SYRTHES 3.4 - User manual

I.Rupp, C. Péniguel

EDF R&D
6 Quai Watier
78401 CHATOU - FRANCE

support-syrthes@edf.fr

2008

Avant-propos

This document constitutes the user manual of the solid and thermal code SYRTHES 3.4. It contains also a guide to use the software in its version coupled to the thermohydraulic code *Code_Saturne*.

Contents

Introduction 1

In a large number of industrial applications [1], thermal phenomena couple fluid and solid. This is typically the case for thermal shocks, when components are subjected to large temperature variation. The resulting expansion may induce mechanical stresses which can lead to cracks. In order to assess the integrity of the components, it is crucial to be able to predict the thermal interactions between the fluid and the solid. Likewise, it can be found very useful to have tools handling both fluid and solid in fields related to heat exchangers.

In some industrial configurations (in air and more generally in presence of gas), heat transfer by radiation cannot be omitted. In fact, this heat transfer mode can become predominant even at fairly low temperature. Not taking into account the radiative aspects can lead to non physical behavior, or at least yield results very far from reality.

For a long time, understanding the phenomena taking place, and optimization have relied on experimental facilities and parametrical studies. Aside their fairly high costs, experimental approaches lead generally to partial information, in a limited number of locations (in fact where the sensors have been placed).

With powerful computers available on the market at very reasonable cost, the numerical approaches become more and more interesting. Numerical tools can help designers and engineers to optimize components and processes. Indeed, numerical tools turn out to be very flexible and well adapted to gain understanding of the phenomena really happening. It becomes easy to realize parametric studies, provided the tool has been well validated against experimental data or analytical formula when they exist.

The organization retained consists in the general thermal code SYRTHES (SYstèm de Résolution THErmique Solide). This thermal solver can be used alone if users are interested only by conduction/radiation problems or coupled with fluid mechanics codes. The document [2] describe respectively the CFD codes *Code_Saturne*. The purpose of the present document is mainly to present the code SYRTHES 3.4 and how it should be used. The procedures one needs to follow, in case the version coupled with CFD codes is required, are also explained in detail.

Some information related to this document 2

The purpose of this document is to make, if not easier at least more agreeable, the use of the thermal code SYRTHES 3.4 and its coupling with CFD codes *Code_Saturne*. Users are kindly asked to consult the user manuals of the fluid code for specific questions relative to these code [2].

2.1 Why this manual ?

The aim of SYRTHES is to be able to handle complex industrial processes. It seems obvious that taking into account thermal phenomena globally can improve significantly the simulations. On the other side, such a global approach induces more complexity since the number of parameters and tools increases. It is easy to understand that solid and fluid thermal properties as well as diverse boundary conditions are then required. In case of a solid thermal coupling, the user needs to consider the global aspects of the problem and not only one system any more. One would like however to reassure users by saying that due to the linear behavior of the diffusion process it is often easier to interpret the phenomena taking place in the solid than in the fluid part.

2.2 For which kind of users has this document been written?

With this manual, we are aiming particularly at the occasional users of SYRTHES having a good knowledge of the pre and post processor, and having followed a short formation on the thermal code SYRTHES 3.4. In case of the utilization of the version coupled with the CFD codes, one supposes that users master these codes as well.

The complete beginners are advised to follow a course (even a short one) to give them some hints on the best ways to tackle the thermal problems with this tool. To solve problems involving fluid calculation, it is easy to understand that a good use of the CFD code itself be compulsory. Users should consult CFD users guide for this phase. It can also be interesting to consult the theoretical manual of this code (see references [2]).

2.3 Organization of the document

This paragraph tends to explicit how this user manual has been organized. It as been divided in several chapter having different goals. The table of contents (at the beginning

of the document), the index, as well as the structure of the document should help users to find the information they are looking for. Summary tables located at the end of this document can also help to either give a direct answer or provide the location where more detailed information can be found.

The document has been fractioned in two large parts:

- **Functionalities and use of** SYRTHES

  The chapter 3 is very general. It focuses on functionalities and underlines some general principles followed by the two developers of SYRTHES. Reading it can prove to be useful for beginners, or for people having questions on the adequation between the possibilities of the codes and the problem they want to treat. The second part of the chapter is important since it is where convention and the methodology used in SYRTHES are given.

  Chapter 4 gives some information on the software structure. This can help users to organize their data to solve the problem they are confronted with. The different files and tools which can be used aside SYRTHES itself are described. Utility programs to process the results to several post-processor format are for example described in detail.

  Chapter 5 is concerned with the different files and the data required for a calculation. Providing the adequate data is indeed a crucial step of a study. Chapter 6 constitutes a detailed review of all possible parameters accessible to users.

  Chapter 7 deals with the users routine. They are very useful to handle complex cases. One should however mention that in a large number of cases, one can treat the problem without having to use any of them, all the information being treated directly through the graphical users interface.

- **Methodology and examples**

  Chapter 8 proposes a possible methodology to do a study. Users can find useful hints on how they could tackle their problems. One describes as well as possible error messages, a possible explanation and then what action should be taken.

  Chapter 9 consists in a collection of several examples treated by SYRTHES. Detailed information and comments have been made. The aim of the two authors was to give a complete description of a typical problem starting from the analysis part, the setting of the different parameters, the calculation itsef up to the post-processing phase. Of course, such a choice induces some repetitions of aspects already discussed in other parts of the document. On the other side, this allows to insist on some physical aspects which have been willingly overlooked in the parameter description part. Users can find there valuable information for their own problems.

Finally, one can find technical description of all the formats used by SYRTHES in the appendix. This can be of particular interest to users willing to use post-processor other than those provided classically (i.e ENSIGHT and RUBENS). Summary tables are also helpful to find quickly information relative to key-words parameters.

2.4 How complete is this manual?

One can find in this manual a complete description of all the aspects that a user can be confronted with when he wants to treat a thermal problem with SYRTHES. In particular, how the software is organized, how to find informations, how to describe their problem and enter the necessary parameters, how to program the user routines and compile them in cases where they are necessary.

It is clear that the purpose of this document is however not to describe the numerical methods used, not even to give sufficient information to extend the functionnalities already implemented in SYRTHES 3.4.For maximum effectiveness, it is assumed that the user performs calculations coupled SYRTHES/ code fluid has the manuals on the use of code *Code_Saturne* [2], [3].

Users interested to know more about the physics and the numerical techniques implemented in SYRTHES can consult reference [4].
Examples gathered in chapter 9 can also contribute to give a better idea of the field of application covered by the code.

2.5 Updating the version

To obtain a better product, users are kindly requested to inform the developing team in case some problems should occur. For this reason, this manual has a page (the troubleshooting form at the end of this manual) that users can fill and send in case of problems. Users are encouraged to fill out this form also if they wish to see new developments or improvement in the code.

Part I

SYRTHES 3.4

Functionalities and specificities 3

In this chapter, we will try to give a precise idea of the potentialities of the code SYRTHES 3.4 and its coupling with the thermal-hydraulic code *Code_Saturne*[1]

One begins to remind the physical phenomena taken into account, then the choices retained to model them are explicited. Finally, the main conventions followed throughout the manual and in the code itself are presented.

Users will therefore use this chapter to:

- check if the problem they intend to solve enters the field of application of this version of the code,
- understand some of the mechanism taking part in the modeling,
- check the convention taken in SYRTHES,
- get some information on the user interface

Finally, one stresses on the fact that it is not question in this chapter to explain how to use a functionnality but simply to put into evidence its existence. The different tasks necessary to use these functionnalities will be treated in the following chapter.

3.1 The thermal conduction

The different possibilities of SYRTHES are now described. One tries therefore to insist upon the advantages and drawbacks of each of them. We wish to warn the readers against a feeling of complexity which could appear from a first reading. Indeed, a great majority of users should only be concerned by only one or at least a small proportion of the potentialities offered. There is for example no need to understand the moving solid aspects if the problem treated is only dealing by a fixed solid with constant properties. The different potentialities have been classified according their difficulty to use them and their probability of occurrence.

[1] Its aim is not however to remind users of the field of application of these CFD codes, which have their own documentation.

3.1.1 The phenomenon simulated

When different part of a body are at different temperature, heat propagates from the "hot" regions towards the "cold" ones. This transfer is made by essentially three means:

- by conduction (heat propagates inside the solid itself)
- by convection (heat is transfered by displacement of one part of this body towards another part of the same body)
- by radiation (heat is transfered by electro-magnetic means, and there is no need for material support any more)

Convective aspects are handled by CFD codes, while the conduction and the radiation aspects (limited to non participating medium for the time being) are treated by the code SYRTHES. The study can be completed by a semi-transparent radiation study (for combustion for example) if the CFD code has such a capacity.
First principles can lead to the following equation for the solid part:

$$\rho C_p \frac{\partial T}{\partial t} = -div\ \vec{q} + \Phi$$

where ρ and C_p are respectively the density and the heat capacity of the material considered. Temperature T is the unknown. The left hand side of the equation constitutes the time dependent aspect of the phenomena, the right hand side of the equation is relative to the way heat behaves and propagates through the material. $\vec{q}$ represents the heat flux, and Φ a volumetric source term.

This equation models the thermal behaviour of a simple material. At the boundary of the medium, several kinds of phenomena can arise either separately or simultaneously. For the modelling of such boundary conditions several choices are offered classically and are detailed further along this chapter.

The previous continuous equation can adopt several forms according the approximations that users are ready to do. In some cases, geometrical caracteristics can lead to restrict the problem dimension to two (cartesian or axisymmetric) instead of three.

3.1.2 Geometrical aspects

Fondamentally, the dimension of space is three. In some configuration, the phenomena behave independently of one of them. To be honest the validity of this approximation is often not exactly verified and is dependent of the user's wealth. Anyway, in that case it is then very tempting to solve the problem in the reduced space. This may reduce dramatically the cost and the difficulty of the study. To stay coherent with the possibilities offered in the CFD codes, SYRTHES can also handle cartesian and axisymmetrical space problems.

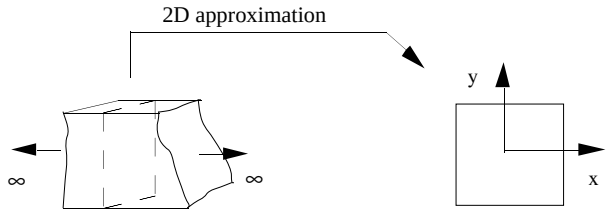

Figure 3.1: Bidimensional approximation

3.1.2.a Bidimensional cartesian

The conduction equation can then be written in a 2D space (x, y). Temperature, physical properties, and boundary conditions, and more generally all elements relative to the case problem treated are dependent of only two space variables. The spatial discretization of the equation is then done on an unstructured mesh generated by the user. This grid generated by mesh generator like SIMAIL or IDEAS-MS contains triangles having six nodes. One underlines that the element's sides are straight lines.

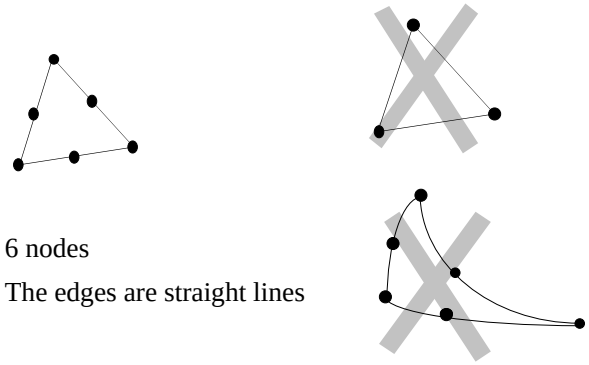

Figure 3.2: Kind of elements used in 2D

3.1.2.b 2D axisymmetrical geometries

Another situation exploits the fact that in some problems axisymmetry is present. It means there is no mean to distinguish one slice from another, both from the geometry point of view and from the behaviour or boundary conditions applied. Thermal phenomena are then calculated only in a slice of null thickness, the third direction being taken into account in the equation itself. Again the reduction from a space having three dimension to a 2D space reduces drastially the amount of calculations one has to perform. The results obtained are as valid as the those obtained by a three dimensionnal approach provided the axisymmetrical hypothesis is valid.

It should be noticed that users can choose the axisymmetrical axis (either Ox or Oy).

Once again the spatial discretization relies on 6 nodes triangles with straight edges.

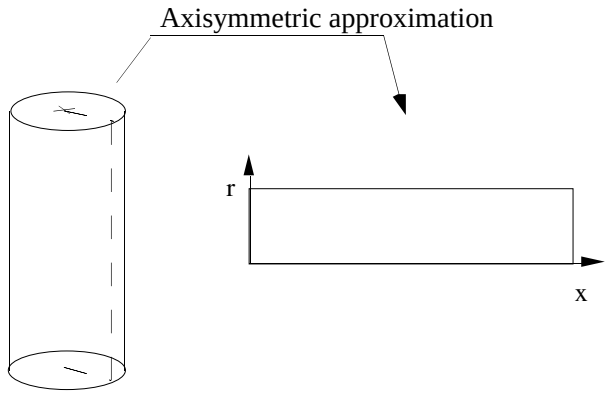

Figure 3.3: Axisymmetrical approximation

3.1.2.c Tridimensionnal

Since the dimension of the space used for solving the problem is compatible with the space of the phenomenon, no restriction is necessary. The spatial discretization is assured by tetrahedron having 10 nodes, and plane faces.
The 3D mesh is provided by users and generated thanks to mesh generator like SIMAIL or IDEAS-MS or any software able to generate the sufficient information relative to the geometry. In the latter case, the format has to be recognized by SYRTHES. (see B for more information).

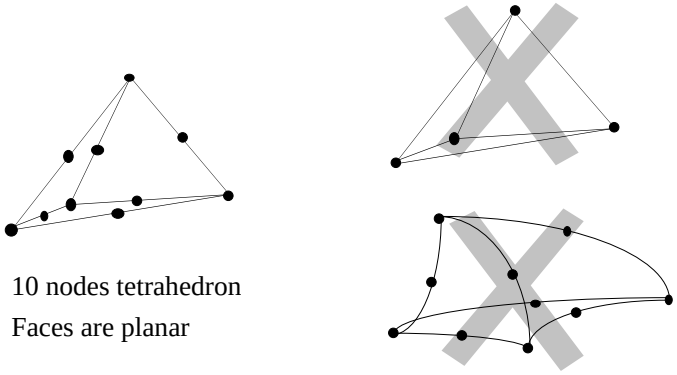

Figure 3.4: Kind of elements authorized in 3D

3.1.3 The material handled

All media transfer heat. However their conductive behaviour can vary considerably from one material to another. One must be able to separate the different material playing a part in a problem. Sometime their behaviour becomes space dependent. This is typically the case when the material caracteristics are temperature dependent. In that case, the temperature equation becomes non linear, but very often, the time variation is sufficiently slow to consider that one can use the local temperature found at a previous time step.

Among the properties which define a conductive material, one finds the density, the heat capacity, and the conductivity.

- $\rho = \rho(x, y, z, t, T, \dots)$
- $C_p = C_p(x, y, z, t, T, \dots)$
- $k = k(x, y, z, t, T, \dots)$

These properties can be defined simply when they are constants on the domain or a sub part of the domain. In more complex situations, a user routine (*cphyso.F*) can be used to implement complex behaviour in any location of the domain.

From a modelling point of view, one relates the flux (a fondamentally continuous quantity) to the temperature gradient through a quantity called conductivity and noted k in SYRTHES. According the material, this quantity can be a scalar, or a matrix. The following paragraphs investigate different possibilities that users may encounter.

3.1.3.a Isotropic behaviour

This is the most frequent case. It corresponds to a solid which if submitted to a point sollicitation will diffuse the information in an isotropic manner (thermal contours will form concentrical circles in 2D and spheres in 3D). Mathematically, this indicates a colinearity between the flux and the temperature gradient. The flux can be expressed with the very classical Fourier law :

$$\vec{q} = -k \ \overrightarrow{grad}\, T$$

The user needs only to define one scalar quantity at each node or element of the mesh. If the conductivity is the same everywhere, then only one scalar is sufficient to describe the conductivity of the material. It is clear that such a choice leads to the cheapest situation in term of memory requirements and computing cost. It is worth noting that it represents a great majority of the problems treated.

3.1.3.b Orthotropic material

In some occasion, heat may not propagate isotropically. Such a medium submitted to a point sollicitation will not lead to concentrical thermal contours. One direction will transfer heat better than another. When the direction of propagation are aligned with the axis of the system of coordinates, one speaks of medium with orthotropic conductivity. Conductivity is then represented by a matrix having the following form :

$$K = \begin{pmatrix} k_{xx} & 0 & 0 \\ 0 & k_{yy} & 0 \\ 0 & 0 & k_{zz} \end{pmatrix}$$

In this matrix, each coefficient (k_{xx} par example), is still allowed to vary in time and space, and more generally may depend of all the local parameters and quantities accessible to users.

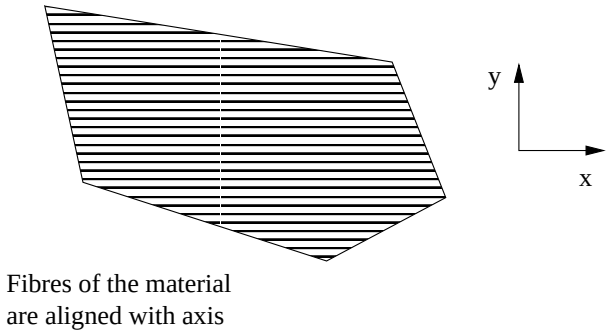

Figure 3.5: Example of material having an orthotropic behaviour

3.1.3.c Anisotropy

This fonctionnality is fairly similar to the two previous ones. It extends the material behaviour to full anisotropy. It means simply that the direction defining the conductive behaviour are not necessary aligned with the axis of the system of coordinates any more. The following figure illustrates a structure whose behaviour could be anisotropic.
The conductivity matrix adopts the following form:

$$K = \begin{pmatrix} k_{xx} & k_{xy} & k_{xz} \\ & k_{yy} & k_{yz} \\ & & k_{zz} \end{pmatrix}$$

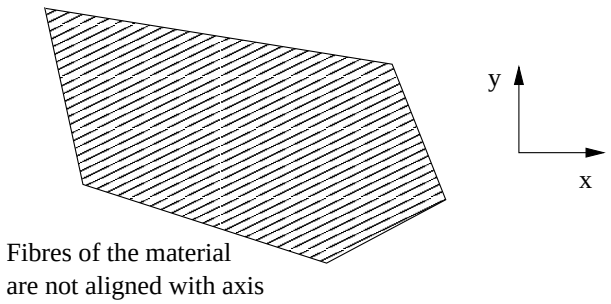

Figure 3.6: Example of anisotropic behaviour

Remark:
This matrix being symmetric and positive, it is easy to prove that there always exists a system of coordinates, in which it is possible to express it under a diagonal form. In SYRTHES, *one uses this property to enter the required data. Indeed, most of the time, users do know the conductivity matrix in a system of coordinates related to the material considered. A user routine allows however to program the most general case possible. It is clear that this last case covers the two previous possibilities. However, the full anisoropic option leads to a much higher cost in term of memory and computing. This is what makes interesting the separation of the handling of the conductivity in three options.*

3.1.4 Initial conditions

The temperature inside the solid domain must be set at initial time t. The temperature distribution can be either continuous or discontinuous, but physically, due to the diffusion operator, a continuous distribution appears very quickly.

Most of the time, the user wishes to impose a constant temperature throughout the domain, or on subdomains defined by references. If the situation is more complex, it is always possible to program a user defined subroutine (*initmp.F*). More details can be found in 5 and 7.

3.1.5 Boundary conditions

In order to solve the problem completly, one has to define the different boundary conditions which apply at the border of the domain. The boundary conditions available in SYRTHES are very classical, and are presented in the following paragraph:

- **Dirichlet** (imposed temperature). One considers that at the border, the temperature stays constant or is a known function of time and space. The value taken by the temperature is known by the user. This condition is relatively simple to handle from a numerical point of view, but it should be noticed that from an experimental point of view (even in a laboratory), prescribing a fixed temperature is generally a fairly difficult task.

  According the case, imposing the temperature may prove to be more or less difficult. If the temperature to be imposed is constant on an identified portion of the border, it will be possible to identify the corresponding nodes by some references at the mesh generator level. Then the keyword related to the Dirichlet conditions will allow to affect the temperature values directly through the interface (see chapter 6). On the other side, if the case is more complex, a special user subroutine is available *limsol.F or limfso.F*. The way to use these Fortran subroutines is detailed in chapter 7.

- **flux**
  Another very common boundary condition is to impose a flux. The flux can be either imposed on nodes or faces. The face choice seems to be better from a physical point of view (cf. 3.5). Unfortunately some grid generators do not have the face information coded internally at the element level (this is typically the case for IDEAS-MS). Then the users confronted to such a situation can use nodes references to impose their flux.

  Again, according the difficulty of the problem, users can use either keywords and the interface to enter the required values, or rely on user subroutines. Very complex cases can be handled, but on the other hand, one has to compile the corresponding programs and link the object file with the SYRTHES library. Chapter 7 give a detailed description of the way to proceed to impose flux thanks to the user routines *limsol.F or limfso.F*

- **heat exchange**
  In many physical cases, the flux is proportional to the temperature difference existing between the temperature surface (noted T) and the temperature of the surrounding medium (noted T_o). Then the flux can be expressed under the form $h(T - T_o)$. The quantity h is generally called a heat exchange coefficient. It is expressed in W/mK. In the case of a forced flow, this parameter is generally related to the local velocity of the fluid as well as the local fluid caracteristics.

  To stay coherent, users will use either the interface or user surbroutines *limsol.F or limfso.F*. Chapter 7 details how to use and program the Fortran surbroutines. It should be noticed that two parameters are required on each node or face of this kind. The first one designes the temperature value (in degree C) while the second one is the heat exchage coefficient.

- **infinite radiation**
  This condition should not be mistaken with "real radiation" from wall to wall in a enclosure treated further. Here it is simply the heat exchange by radiation (gain or loss) between the wall and the surrounding space.

- **symmetry**
  In many studies, the domain of calculation can be advantageously reduced if it presents some symmetries. Then the computation can be done on only 1/2, 1/4, (or 1/8) of the domain. From the conduction point of view, a symmetry condition is equivalent to specify zero flux on the boundary. This condition does not require any parameter. It will be sen later that on the other hand, users have to indicate some extra information if radiation from wall to wall is included.

- **periodicity**
  Periodic boundary conditions can apply between faces having any orientation. The possible geometrical transformation being either a translation or a rotation. The following figure illustrates how to handle a problem on a reduced domain thanks to a periodicity of rotation :

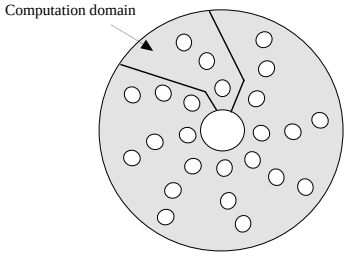

Figure 3.7: Periodicity of rotation

It should be noted, that it is possible to handle several directions of periodicity simultaneously (up to 2 in 2D, and 3 in 3D). This allows to treat easily and exactly the case of a very large plate having repetitive pattern.

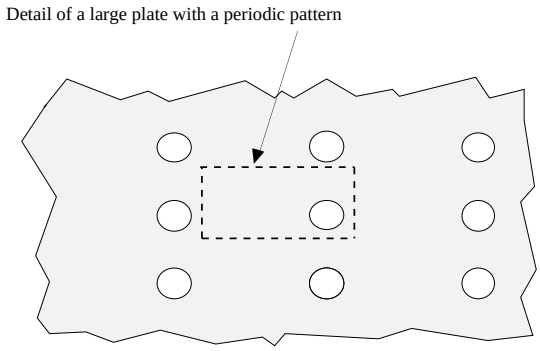

Figure 3.8: Application having periodicity in 2 directions simultaneoulsy

In the above example, the reduced domain requires to specify two directions of periodicity :

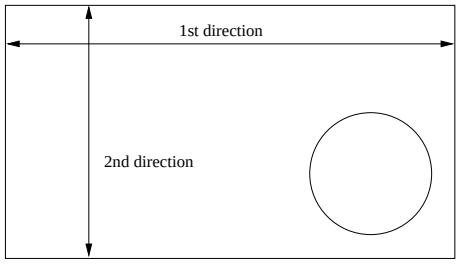

Figure 3.9: Application of periodic case in 2 directions

One should be conscious that in such a case, the corner nodes are very particular. Indeed, each of them is submitted to the two periodicities; then the contribution of the surrounding elements coming from 3 directions has to be taken into account. This principle is exposed in the following figure.

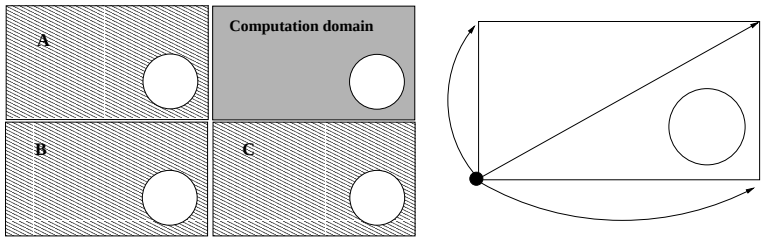

Figure 3.10: Special treatment of the corner nodes

In 3D, the principle stays identical, but then a corner nodes needs the contribution of 7 domains.

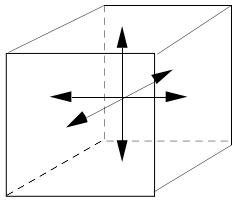

Figure 3.11: Periodicity along 3 directions

3.1.6 Volumetric flux

Sometimes, physical phenomena leads to make heat appear inside the solid itself. This is typically the case for metallic pieces submitted to electromagnetic phenomena. The resulting Joule effect can be modelled thanks to source terms.

SYRTHES allows to impose source terms (or volumetric flux) on nodes or elements of the domain. These source term can be time and space variable if necessary. In case of constant volumetric source, user can very easily impose them through the user interface (see chapter 6). For more complex situations, the user subroutine *cfluvs.F* allows to program a time or space dependent source flux.

3.1.7 Contact resistance

In some industrial mecanisms, often pieces belonging to a system are composed of different materials. These materials are often either glued or bolted together, and heat can transfer. A more precise study shows that even if optically well in contact, the two materials in contact cannot be considered as forming a continuous solid. It may happen that a small gap of air leads to a discontinuous temperature field. The heat flux stays of course a continuous notion.

It may be useful to use this potentiality if one wants to simulate a small default in a solid (a thermal representation of a crack). It is not reasonable to consider the piece as continuous, but is also impossible to consider a total independence between the two sides of the gap, indeed the heat flux keeps flowing through the gap.

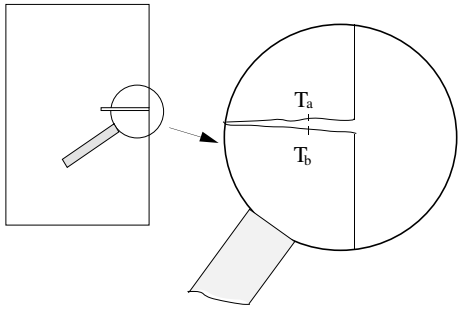

Figure 3.12: Contact resistance

It is then possible to introduce the notion of contact resistance. It is in fact a heat exchange condition of a particular type. The external condition is constituted by the temperature of the face located on the other side of the gap. Unlike boundary conditions previously described, temperatures of both faces in regard stay unknowns of the problem and are found as results of calculations. They are likely to vary at each time step.

One has the following relationships :

$$\begin{cases} g\,(T_a - T_b) = k_a\ grad\,T \\ g\,(T_b - T_a) = k_b\ grad\,T \end{cases}$$

where T_a and T_b are unknowns of the problem.

One can use either the interface described in chapter 6, or the user subroutine *limsol.F* (see chapter 7) to set the proper value of the contact resistance.

Word of Caution
In pratice, the determination of the coefficient g may prove to be delicate. It is easy to understand that a certain portion of empirism must be introduced since it caracterizes the way two materials are in contact. The use of this fonctionnality requires therefore a certain knowhow.

3.2 Thermal radiation

3.2.1 Generalities

All susbtances emit electromagnetic radiation on a large band of frequencies. This radiation is related to the internal energy of the body. The higher the internal energy the higher the electromagnetic agitation, leading to the emission of ultra-relativist elementary particles. Inversely, the energy carried under electromagnetic radiation can excite the electrons in the medium and therefore increase the system internal energy.

This mode of heat transfer is quite different from the convection or conduction ones. Indeed, there is no need for a medium to propagate radiation[2]. Instead of a simple flux vector[3] as for conduction, the radiative flux corresponds to the contribution of radiation coming from all the directions of space. This leads to an integral formulation. When the three heat transfer modes are coupled together, the equations to be solved are integrodifferential ones. These kind of equation are often considered as very difficult to handle.

In an enclosure, complex heat exchanges are present when radiation leaves one patch to reach another location of space where it is partly reflected and emitted, a great number of times.

Fortunately, in numerous situations, simplifications are possible, while staying rigourous. The choices and approximations used for the treatment of radiation in SYRTHES are presented below.

- for the time being, only radiation from wall to wall is considered. It means that the intermediate medium does not interact, or can be considered as transparent.
- the solid bodies are supposed to be opaque
- the behaviour of the walls is supposed to be diffuse
- the behaviour of the walls is supposed to be grey (at least by band)

For more details on these concepts can be refer to the reference [5].

3.2.2 The treatment of thermal radiation in SYRTHES

Thanks to the different approximations, often justified in most cases, the problem can be discretized in time and space. This leads to the very well known method of radiosity. Then the following system of equation has to be solved :

[2] Radiation propagates very well in the vacuum, fortunately for life on earth.
[3] which leads to the notion of differential equation.

$$\begin{pmatrix} 1-\rho_1 F_{11} & -\rho_1 F_{12} & \cdots & -\rho_1 F_{1N} \\ -\rho_2 F_{21} & 1-\rho_2 F_{22} & \cdots & \vdots \\ \vdots & \vdots & \ddots & \vdots \\ -\rho_N F_{N1} & \cdots & \vdots & 1-\rho_N F_{NN} \end{pmatrix} \begin{pmatrix} J_1 \\ J_2 \\ . \\ J_N \end{pmatrix} = \begin{pmatrix} E_1 \\ E_2 \\ . \\ E_N \end{pmatrix}$$

In the pevious system of equations, E_i is the emitance of face i and ρ_i designates the reflectivity (with $\rho_i = 1 - \varepsilon_i$, ε_i being the emissivity of the face i).
The unknowns are the radiosity[4] noted J_i in the previous system, leaving each of the N faces composing the mesh of the enclosure. A purely geometrical quantity[5] noted F_{ij} appears in this method. Physically it can be interpreted as the proportion of energy leaving the face i which reaches the face j. One has :

$$F_{ij} = \frac{1}{S_i} \int_{x \in S_i} \int_{y \in S_j} \frac{cos\theta_1 \, cos\theta_2}{\pi r^2} \; V(x,y) \; dy \; dx$$

with S_i the surface of the face i, x and y being two points belonging to the faces i and j. θ_1 and θ_2 are the two angles between the normals of each face and the line of sight between the two point x and y. r is the distance between point x and y, $V(x,y)$ is a function of visibility which indicates if point x can see or not point y. This quadruple integral is often very hard to calculate. Users interested can refer to reference [5] to get more details on possible ways to calculate such a quantity.

3.2.3 Validation

The handling of thermal radiation has been validated on a certain number of configuration.

A first step has been to validate precisely the calculation of the view factors, which constitutes a key point in the radiosity method. Elementary comparisons have been made against analytical expressions for simple cases where analytical solutions do exist. Then more complex situations including cases where occluding faces exist have also been studied.

In a second part, tests investigating the solver used to solve the system of equations have also been done. Again the solutions found with SYRTHES have been checked against solutions coming form the literature. In all cases, very satisfying results have been obtained. Users can refer to reference [6] to get more details on these aspects.

3.2.4 Geometry

As for conduction, SYRTHES can handle radiation in 2D cartesian, 2D axisymmetrical and 3D cases.

The treatment of axisymmetrical configurations has given way to special developments which prevent to recreate a tridimensional mesh for the view factor calculation. In par-

[4] One recalls that the radiosity is the flux of radiation which leaves the considered face
[5] This quantity is often named form factor or view factor

ticular, the data base stays always 2D even if all calculations take into account the 3D effect.

In some other applications, it can be interesting to reduce the domain of calculation by taking advantage of symmetries or periodicities. The radiation fonctionnalities implemented in SYRTHES can deal with these configurations. Multiple symmetries are allowed (up to 2 in 2D, and 3 in 3D). However users should pay attention that symmetries do necessarily lead to closed domain. In particular two symmetry planes facing each other are not possible to handle. Indeed, in that case, the domain whould virtually extend to the infinity.

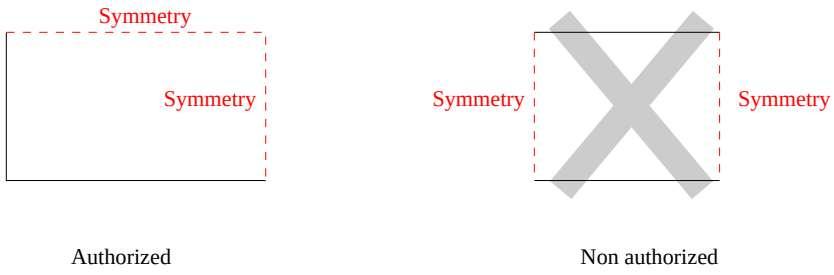

Figure 3.13: Symmetries for the radiation grid

Concerning the periodicity, it is restricted to a periodicity of rotation (the only one leading to closed domain). The angular section provided by users must lead to 360 °when the initial pattern is duplicated a certain number (1/2, 1/3, 1/4, 1/8, ...) of times.

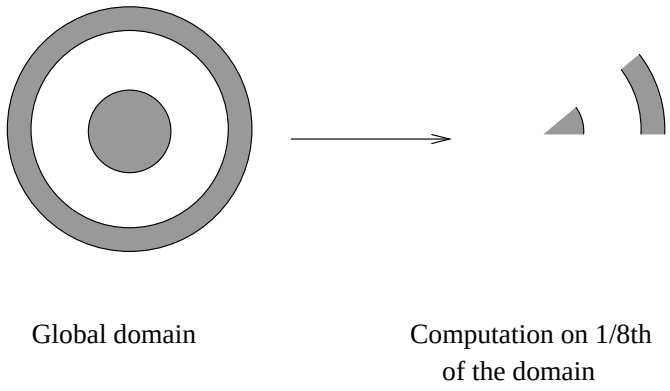

Figure 3.14: Periodicity for radiation

3.2.5 Physical properties

SYRTHES can handle radiation for bodies grey by bands. Users can define several spectral bands and give the spectral emissivity for each of the them. Emissivity can vary with space, time, etc... if necessary.

3.2.6 Boundary conditions

For the radiation implemented in SYRTHES, the logical situation is to be in contact with a solid body in which conduction is solved. However, some situation can arise in which it seems interesting to propose other kinds of boundary conditions. The most frequent case corresponds to situations where the grid used for radiation does not define a closed domain. This can be the case, when inlet or outlet are present.

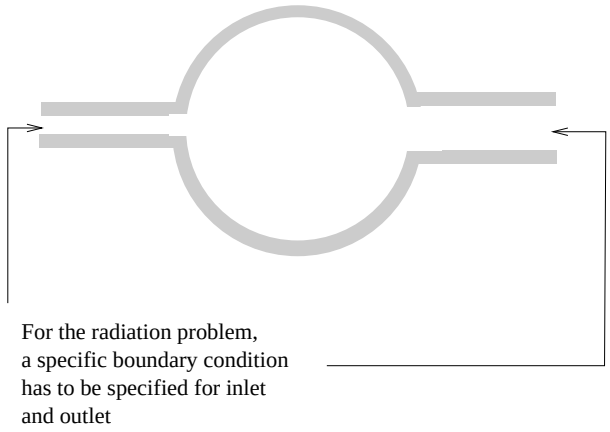

Figure 3.15: Boundary conditions possible for radiation

It is possible to set the following boundary conditions :

- **coupling with conduction**
  This is the condition that most radiation patches should be affected with.
- **imposed temperature**
  This is the condition which is generally used to close the computational domain of radiation,
- **imposed flux**
  In multi band problems, the flux has to be provided by band.

3.3 Coupling with CFD codes

As seen in the introduction, our aim is also to propose a numerical tool, as flexible as possible, to handle themal coupling between fluid and solid domain. Taking into account the thermal coupling has many interests which are described below.

3.3.1 A better handling of boundary conditions both for fluid and solid sides

When doing a numerical simulation of a phenomenon, one has of course to solve the equations modelling the phenomenon inside the domain of interest but also to set proper boundary conditions at the interface. Most of the time boundary conditions from the thermal point of view are difficult to grasp. Taking into account the solid domain, can in

many occasions, reduce this difficult task. For example, when a pipe is thermally insulated on its outer side, imposing an adiabatic condition outside is quite rigourous. On the other hand imposing an adiabatic condition at the fluid/solid interface is clearly wrong.

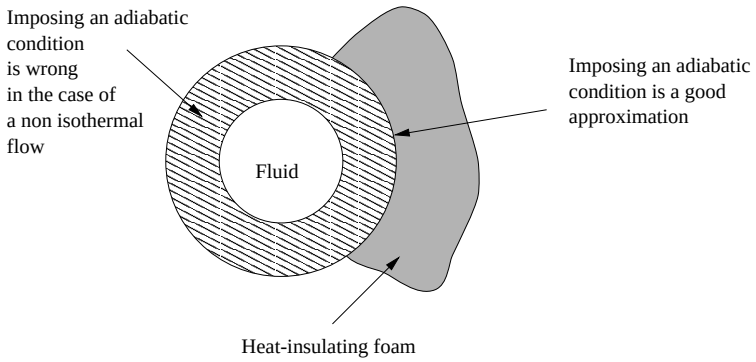

Figure 3.16: Non isothermal flow in a pipe insulated

In industrial configurations, it is often possible to have access to external boundary conditions, but it is often very costly, if not impossible to have access to information located inside the fluid or at the fluid/solid interface.
Likewise, it is often much easier to place a thermal captor on the external side of a body, than inside to validate a calculation or a process. From a validation point of view, one can hope that if the thermal comparisons are satisfying on the external side of the wall, it suggests that the flow has been well calculated, and the transfer through the wall has been correctly handled as well.
Sometimes, a source term is present inside the wall (Joule effect for example), in which case it whould be very difficult to simulate such a behaviour at the fluid/solid interface.

3.3.2 Thermal shocks

The thermal interaction can be fundamental in case of thermal transients, which are very frequent in industrial processes (nuclear industry for example). Here we take the case of a thermal shock (severe increase of the fluid temperature at the inlet) in a piping system. The thermal inertia of the solid will lead to a gradual increase of temperature at the wall, and inversely a cooling of the near wall fluid. This may lead after a certain distance to reduce quite considerably the steepness of the shock. At the end of the pipe, the thermal load experienced by the component under study is much less severe and can become compatible with safety requirements, unlike the very conservative attitude of wall without thermal inertia.

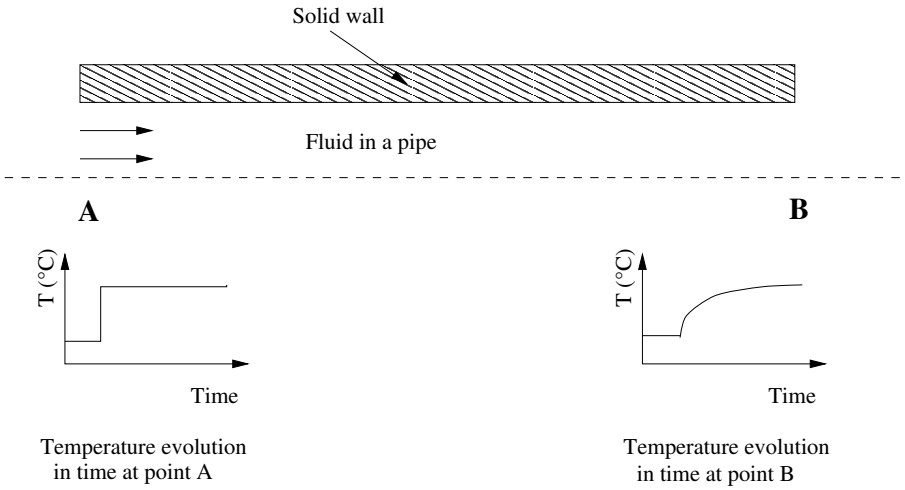

Figure 3.17: Reduction of the severity of a thermal shock due to wall inertia

3.3.3 Accessing to the solid thermal wall

The interest of the fluid/solid thermal coupling can also come from the knowledge of the solid thermal contours it can provide. In that case, the fluid calculation is meant to provide better boundary conditions for the solid calculations. This can be the case in cooling processes, when metal plates are cooled down by jets of air, or by free convection. The classical approach consists in guessing the effect of the fluid through heat exchange conditions. Unfortunately, finding the proper coefficient is a very difficult task, and it is very easy to do mistakes, since the thermal flux going through the interface depends of local local quantities like the velocity, and the temperature.

Once the solid thermal field determined,

- it becomes possible to analyse how efficient the cooling device is, and how it could get optimized.

- it becomes possible to determine local hot, or cold spots. Indeed, it is often preferable to prevent the appearence of hot spots which may lead to damages or disfunction (this is typically the case in electronic industry).

- Finally, it can be interesting to calculate the mechanical stresses (with an idependent mechanical code) induced by a non uniform temperature distribution. A utility program, *syrthes2med*, allows to project the temperature field given by SYRTHES on a grid accepting all kind of elements which can then be processed by a mechanical code (see page 43)

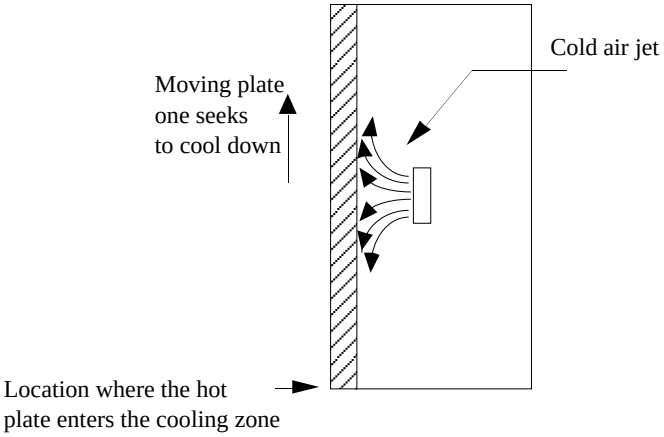

Figure 3.18: Example : cooling of a moving plate

3.4 SYRTHES : A code based on modularity

One tries in this paragraph a feeling on the choices retained for the organisation around SYRTHES as well as the motivations behind. We will insist upon the potentialities and the flexibility provided by these choices when handling a problem.
This chapter will also revisit the main specificities of SYRTHES as well as conventions like (references, periodicity, etc...) to explain the concepts and choices retained in the code.

3.4.1 Coupling SYRTHES / CFD codes

Let's consider a fluid flow in contact with a wall. The fundamental principles of mass, momentum and energy conservation lead to general relations. If one is particularly interested by the thermal phenomena, several regions (illustrated in the next sketch) can be distinguished.

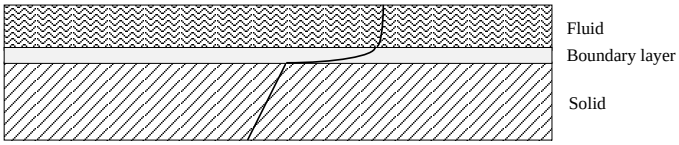

Figure 3.19: The different regions to take into account

3.4.1.a The fluid region

One is here interested in a region purely governed by fluid phenomenon. According the case, a turbulence modelling may have to be used. The use of a coupling with a solid does not modify the equation solved. Users will have to refer to the documentation relative to the fluid code used, to have a good understanding of the limitations and potentialities of the model implemented.

3.4.1.b The fluid boundary layer

In that region special phenomena take place. Indeed, models have to take into account turbulent diffusion, molecular diffusion which play more or less according the distance to the wall and the kind of flow considered. The gradients of the unknowns are often considerable, which makes them difficult to handle numerically. Classically, instead of solving "exactly" the equations valid in that region, one prefers to model the phenomena by a function of transfer between the solid and the fluid.

Among the different options, the modelling used by the fluid code has been retained. Such a choice is motivated first to be coherent with the modelling already existing in the fluid (more details can be found in [7]), and secondly due to the fact that this region is before all a fluid region and therefore stays outside of the domain handled by SYRTHES.

3.4.1.c The solid region

The temperature is governed by a diffusion equation and by the radiation phenomena. The potentialities have been previously described.

3.4.1.d Geometrical decoupling, computing et numerics

Among the possible solutions to handle the global problem, i.e. the solving of the thermal equation in the fluid and solid domains, extending the thermal equation from the fluid to the solid could have been proposed. Indeed, fundamentally, one only needs to eliminate the convective term. For some moderately complex problems, one can generate only one mesh for both domain, i.e. only one data base (nodes and elements). Then only one big system of equation whould have to be solved. The solid nodes whould be affected by the special condition of zero velocity.
Such a method has however numerous drawbacks :

- **this option is not optimal from a memory cost point of view,**
- **it is not possible to have different degrees of spatial discretization,**
- **generating a grid can become very difficult when tackling complex 3D geometries,**
- **it obliges to do major changes inside the fluid code,**
- **for each new fluid code, the work has to be done again.**

Owing to these remarks, it has been decided to conceive SYRTHES as decoupled as possible from the CFD codes. The only obligation, which seems natural, is that both domains approach as well as possible the same geometrical interface.
Solvers are also completly decoupled, which implies that it is possible to use the numerical method the most adapted in each domain. From a programming point of view it constitues also a good way to minimize the introduction of bugs in the CFD code.

3.4.2 The treatment of thermal radiation

The radiation part can almost be considered as an optional module of SYRTHES. Again, one has decided to decouple, from a geometrical and programming point of view, as much as possible the treatment of radiation from those of conduction.
The radiation module implemented in SYRTHES uses an independent grid provided by users. This mesh contains two nodes segments in 2D and three nodes triangles in 3D.

The choice to decouple the spatial discretization has several advantages.

- it is possible adjust the degree of refinement for radiation and conduction independently. It is interesting to recall that the radiosity method retained may become quite memory demanding if the number of independent faces becomes large. Indeed for a surface mesh of N independent faces, one needs to store $N(N+1)/2$ view factors. It is therefore important that a need in a local refinement for the conduction does not imply the same refinement for the radiation grid, which whould lead to a prohibitive number of view factor. An example is given on figure 3.20.

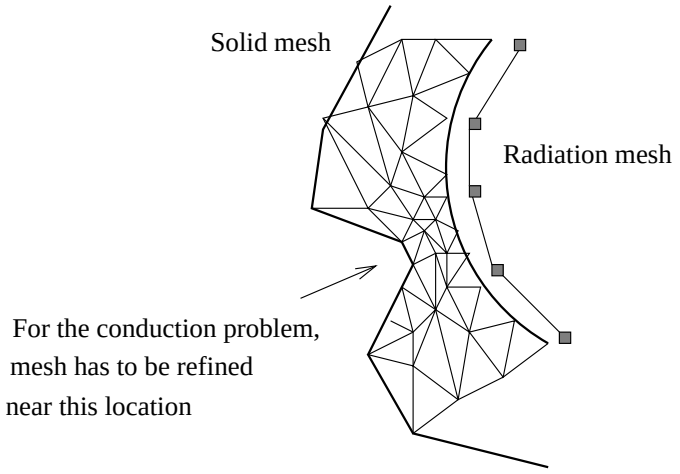

Figure 3.20: Independent grids for radiation and conduction

- The use of a specific grid for the radiation allows to take into account special boundary conditions. Indeed, in some cases, one needs to close the mesh and impose a constant temperature on these faces. This is the case when inlets and outlets are present as in figure 3.21. In this sketch the radiation grid is represented by dotted lines. This figure shows why it is necessary to extend the radiation grid to take into account the inlet and the outlet. On the faces, a special condition will be applied.

The radiation exchange are solved on this grid and then transfered towards the conduction grid, where they are used as boundary conditions.

3.4.3 The flexibility provided by the choices retained

The numerical and geometrical decoupling prevent the drawbacks presented previously. Among the advantages, one finds :

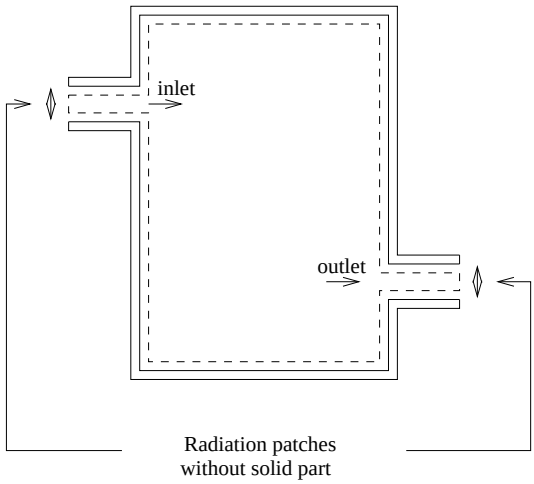

Figure 3.21: Handling of the inlet and outlet faces

- **an optimization in terms of memory and CPU cost,**
- **a great flexibility when doing a study.** Fluid and solid grids can be genererated by different teams,
- **it becomes easy to provide extensions towards other numerical codes.** For example mechanical codes which only requires the temperature distribution inside the solid.
- **it becomes possible to propose to couple** SYRTHES **with other CFD codes, which whould be interested by such a functionnality.**
- **it allows to minimize the maintenance and the evolution costs, indeed each code can be modified independently.**
- **it ensures that** SYRTHES **will keep existing, whatever the evolution of one particular CFD code.**

Among the drawbacks, one could say that the number of data files to generate and result files obtained increases, but this is quite logical, if one considers that the simulation takes more phenomena into account. Moreover the code organisation should limit the problems.

3.5 The use of references

3.5.1 References on a conduction grid

Most grid generators offer the possibility to impose references (or color) on nodes and/or edges and/or faces of the meshes.
SYRTHES uses these references to identify a group of nodes or faces subjected to conditions of the same kind. At the solid level, users has the choice to impose boundary conditions like flux, heat exchange, (sometimes coupled to a fluid) on nodes or faces (cf. 6.2.4). It

should however be noticed that only faces allow to set discontinuous boundary conditions. Regarding the Dirichlet boundary condition, it is always imposed by nodes (whatever the choice retained for the other conditions).

The following figure presents a case where boundary conditions by face can be interesting. The discontinuity at the edge level can be taken into account. According the side from which it is seen, the edge will be affected either by a flux or another.

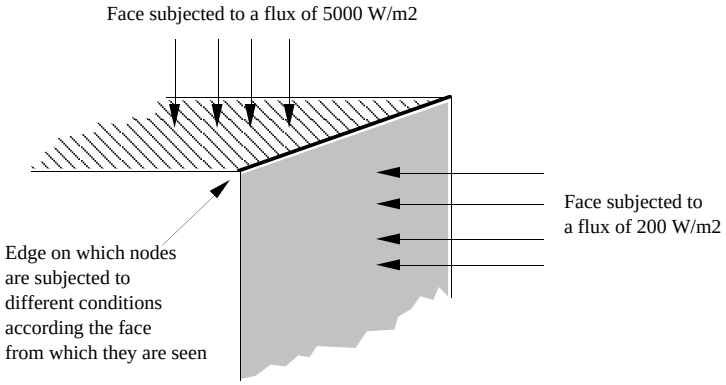

Figure 3.22: Advantage provided by boundary conditions set on faces

References are compulsory to identify and calculate some numbers relative to :

- the different kind of boundary conditions (Dirichlet, flux, heat exchange, contact resistance, infinite radiation),
- volumetric flux (source terms),
- the locations where periodicity will take place,
- the solid part coupled with a fluid,
- the solid part coupled with radiation,
- the fluid part coupled with conduction.

The references can also be used to identify different materials. In that case, it is possible to use references on the nodes or the elements. The latter seems more logical et allows to take into account the discontinuity between two materials in contact. It should be noticed that distinguishing the materials by using different references is not compulsory, since users can rely on user subroutines, but in practice it is advised to use element references .

Likewise, a group of nodes subjected to the same kind of boundary condition can be given the same references.

The references are necessarily between 1 and 99 (at least for the version 3.1 of SYRTHES).

Becareful:
A node having the reference 0 is considered as a node without reference.
Note from the authors
Even if it is always possible to rely on user subroutines, using references as much as possible is quite advisable. Indeed, it leads generally to a better structuration of the grid and make a lot easier the calculation description. This is particularly true when the diverse conditions are constant in time: Then users can describe entirely their problems through the interface without having to do any FORTRAN programming.

3.5.1.a The references in SIMAIL : choices and conventions

When the grid is generated by SIMAIL, the references and sub-domain numbers can be used directly by SYRTHES. Each node has the reference given by the user; each middle node has the reference of the edge to which it belongs. Concerning the face references, they are used only when this option is explicitely specified by users. (cf. key-words BOUNDARY CONDITIONS BY NODE OR BY FACE). The convention retained is the following : In 3D, the face reference, as it is defined inside SIMAIL, is used directly. In 2D, the edge is considered as a face, therefore the edge reference is used as face reference). The numbers characterizing the sub-domain (element reference) is used only if this option is explicitely wanted by users ((cf. interface **'Set up of the material properties by'**).

3.5.1.b The references in Ideas : choices and conventions

Concerning the references inside IDEAS-MS (or colors), the dataset included in the Universal files have only nodes and element references. The notion of face or edges is not included. In case users wish to use face boundary conditions, a user subroutine has been put at their disposal (*inrefa.F*). With that subroutine, it is possible to define a face reference starting from the nodes references composing that face. Users interested by such a possibility can refer to paragraph 7.6 to get a precise description of that subroutine.

3.5.1.c The references coming from a SYRTHES file: choices et conventions

The files having the format recognized directly by SYRTHES are generally issued from a previous calculation. They can also be generated by users having mesh generators other than SIMAIL and IDEAS-MS whose internal format is not yet recognized directly by SYRTHES. The file contains references put on the nodes and elements and faces if this information is available.

3.5.2 References on the radiation grid

In the case of radiation, the mesh is a surface (i.e composed of triangles in 3D, and edges in 2D). SYRTHES uses only the element references.

One has to recall that in IDEAS-MS on speaks of "element color" while in SIMAIL it is question of "color of sub-domain".

3.5.3 References of nodes, faces, or elements ?

The following table summarizes the kind of references possible to set according the type of condition to impose.

Boundary condition Dirichlet	node
Boundary condition flux	node or face
Boundary condition heat exchange coefficient	node or face
Boundary condition contact resistance	node or face
Boundary condition infinite radiation	node or face
Solid zone coupled to the fluid	node or face
Solid zone coupled to the radiation	node or face
Fluid zone coupled to the solid	face
Periodicity	node
Volumetric sources	node, element or nodes by element
Material properties	node, element or nodes by element

3.6 Initial conditions, physical properties and boundary conditions

The feedback we are getting from users shows that in most problems, the conditions used stay "simple". For example, initial conditions are most of the time uniform on all the solid domain, or at least by block, and user boundary conditions are often constant in time.

To improve the ergonomics of the code, it has been decided to let users enter their data directly through the interface when their problems are fairly standard. In numerous calculations this prevents to do any programing. Moreover it becomes then possible to change boundary conditions instantaneously.

This method applies when the boundary conditions and physical properties are constant in time, and spatially constant by block. Then setting the proper values relies on the references previously defined on the mesh.

It should be noted that if defining the reference is compulsory to identify the kind of boundary conditions (cf 3.5), the setting of the numerical values does not necessarily use them. Typically user subroutines can be used to handle specific behaviour.

3.6.1 General principle to use boundary conditions

Setting boundary conditions has two phases :

1. Give the list of references identifying the nodes or faces subjected to that particular condition. This is a compulsory step. It allows SYRTHES to set up the proper inner tables.

2. Give the adequate numerical values corresponding to the references. This step can be done directly through the interface or via user sub-routines for more complex cases.

It should be noted, that for a given condition, the instruction are treated in the order in which they have been defined. If a conflict or an ambiguity exists, then the condition defined last will be the one applied.

Let's also underline that if conditions are decribed both in the interface and in user sub-routines, the conditions progammed in the FORTRAN routines always have the priority.

The way to enter the calculation parameters is described in chapter 6. In all situations, the nodes or faces for which no value has been given (physical, boundary, initial, etc...), will be affected by the default values. (see chapter 6).

An example of boundary conditions (here flux) through the interface is given below.

```
'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   2 5 12

'CLIM'    'DIRICHLET'      25.          2
'CLIM'    'DIRICHLET'      40.          5 12
```

In order to set a condition, the user is always asked the list of references on which that particular condition should be applied. A list limited to the negative value "-1" means that this condition is to be applied to all objects belonging to this condition.

```
'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   2 5 12
'CLIM'    'DIRICHLET'      25.         2 5 12

'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   2 5 12
'CLIM'    'DIRICHLET'      25.         -1
```

3.7 Conventions for the different units

All quantities are expressed in the international system of units.
The only exception is the **temperature** which is always expressed in **degree Celsius** (units which is often easier to grasp for users).
One underlines the fact, that each time the temperature is available in user subroutine to define physical quantities (like the density, or conductivity) SYRTHES is providing the

temperature in degree Celsius. In case some laws of variation requires a temperature expressed in Kelvin, the user has then to do the transformation (add 273.15) himself.

Appendix A summarizes most units encountered in the code.

3.8 Definition of the angles

When entering some parameters, the user may have to provide a certain number of angular quantities. **All angles used have to be defined in degree**.

On the other hand, for the moving solids, users have to define a rotation velocity **expressed in rad/s** (this choice is motivated by the will to follow the conventional habits).

Often the user is required to define 3 angles which represents respectively the rotations around the 3 axis x, y and z. The conventions retained are described in the following figures.

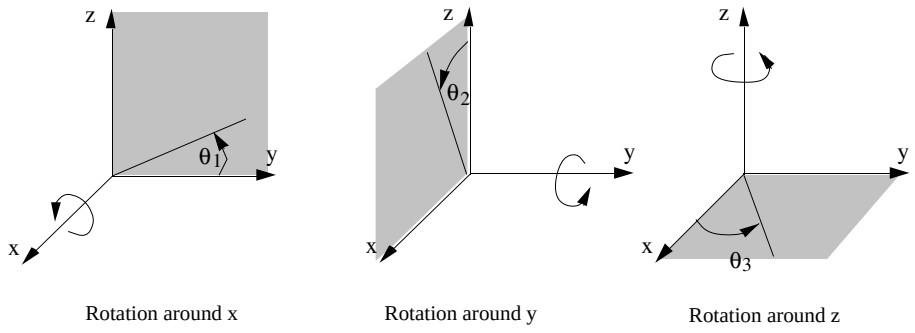

Figure 3.23: Conventions for the angles

3.9 Some information regarding the code SYRTHES 3.4

This version has been written in FORTRAN-77 and C. This allows the code to be fairly platform independent.

The two authors have always tried to respect some development criteria to improve the quality of the code. For example the FORTRAN instruction IMPLICIT NONE is compulsory.

A validation phase of SYRTHES has been performed. The choice retained has been to compare the numerical results obtained against analytical expressions.

The numerical methods retained by the two conceptors are fairly robust and fast. They allow to handle problems having several hundreds of thousands of nodes on workstations,

and several millions of nodes on mainframe like CRAY-C98. The reference [8] give some statistical information on both mainframe and workstation.

3.10 Perspectives

The functionnalities brought by SYRTHES 3.4 and its possible coupling with CFD codes should allow the simulation of a large range of problems. Special extensions are always possible, if users ask them to be developped.

Among the possible extension, one has in mind the implementation of the radiation phenomena in semi-transparent medium, like in glass. It is likely that solar radiation will be integrated soon.

It is possible that some developments to take into account coupled transfer (temperature, humidity, air) appearing in buildings will also be implemented.

Finally, it is interesting to note that due to the decoupled structure of SYRTHES, it seems easy to extend the coupling of SYRTHES towards the electromagnetism or other fluid or radiation codes.

A prototype (using CALCIUM [9][10] and PVM[11]) has been tested by the authors, it allows to perform fluid and solid calculations on different processors.

In the same spirit, this prototype allows SYRTHES to manage the coupling with several fluid codes simultaneously. This make accessible the simulation of very complex systems like heat exchangers, where several flows with very different characteristics are present simultaneously. Illustrating cases [12] [13][10] where SYRTHES-*Code_Saturne* interact simultaneoulsy have been performed and show that multiphysics problems can be quite easily handled with such an approach.

Finally, thanks to utility programs like *syrthes2med*, it is possible to transform the results obtained from a SYRTHES calculation in data directly accessible to mechanical codes (notably the EDF mechanical code CODE_ASTER) to post-calculate the mechanical stresses induced by the temperature distribution calculated by SYRTHES.

Global organization 4

This chapter wants to give a global view of the file organization in SYRTHES, and for the version coupled with a CFD code.

For both the standard version of the code and the coupled version, the philosophy stays the same. It is obvious that the coupled version is more complex since it requires simultaneously the fluid and solid grids, as well as the data files describing the medium characteristics in each domain. Basically, twice as much information is necessary.

However, as previously mentioned in this document, a fairly decoupled structure has been retained. This appears quite clearly at the user level. It is easy to identify the portion relative to the fluid and the solid. This should facilitate greatly the understanding of this coupled numerical tool for new users, and for users having a previous knowledge of either the fluid or the solid part.

Taking into account the radiation consists only in adding a additional module. Such an approach allows to clearly estimate the influence of the corresponding heat transfer modes. The organization of the conduction part is not more cumbersome, only one keyword has been added. It enables the radiation functionality.

In case the radiation has been activated, the user must provide complementary data, like additional grids and a data file specific to radiation.

Such a modular approach is particularly flexible when one wants precisely estimate the influence, or the effect of radiation in a given problem. Starting from a problem where conduction + radiation have been taken into account, one only needs to deactivate the keyword setting the radiation calculation to perform a simple conduction study.

4.1 Organization of the SYRTHES files

One presents here the different files which play a part in a study involving thermal conduction in a solid. The complete description of these files is given in chapters 5, 7 and appendix B.

4.1.1 Thermal conduction calculation

The sketch 4.2 presents the general organization of the data and results files for a conduction calculation.

4.1.1.a The data files

The input files to be given are:

- a geometric file containing the unstructured mesh of the solid domain. This files contains the list of nodes and their coordinates, the connectivity of the elements, as well as the nodes and element references. Paragraph 4.2.1 gives information on the possible tools to generate such a file.
- a file with keywords describing the problem to be treated, the options retained, the calculations and numerical parameters, sometimes the physical characteristics when they are constants, etc...
- FORTRAN user subroutines useful when very complex problems or behavior have to be described. Typically, when the material properties or boundary conditions vary with space and time.

4.1.1.b The results files

The results files present in a classical thermal conduction study are given below :

- a **geometric file** which contains the geometry of the solid domain (required for the post-processing step). This file is also compulsory when the problem deals with moving solids, since the coordinates of the nodes are changing during the calculation.
- a **result file** which contains the temperature at each node of the solid grid.
- a **chrono file** similar to the previous one, except it contains the results at several time decided by the user.
- a **probes result file** which contains the temperature evolution on a limited number of nodes. This option can be very useful for people interested by thermal transient, or the heating or cooling of several specific spots.

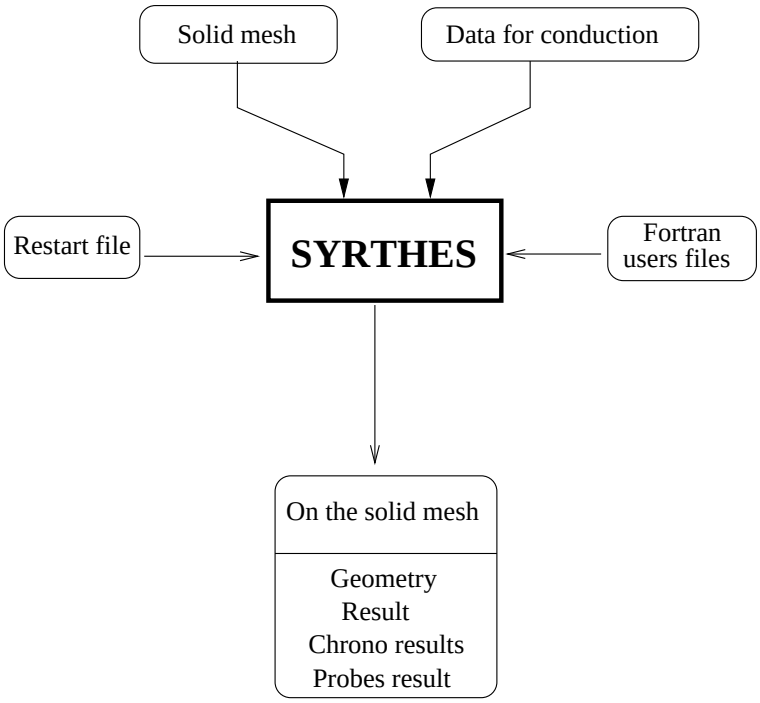

Figure 4.1: Organization sketch

4.1.2 Conduction + radiation calculation

The previous results file are not affected but are completed by a certain number of files specifically related to the thermal radiation phenomena.

4.1.2.a The radiation input files

There are two radiation input files:

- the radiation grid
- a parameter file generally noted *syrthes.ray* describing the case by means of keywords. It is very similar to the file *syrthes.data* used for conduction.

  One underlines that the only authorized radiation elements are 2 nodes segments in 2D and 3 nodes triangles in 3D.

4.1.2.b The radiation result files

Taking into account the radiation in the calculations do not necessarily generate specific radiation files. Indeed, the change are observed on the solid temperature. Therefore, the traditional conduction result files are sufficient to analyze the coupling of the radiation+conduction phenomena. However, the authors have considered it could be interesting to provide direct access to some specifically radiative quantities. Therefore, it is possible to ask SYRTHES to generate some results directly on the radiation mesh.

As for the solid, one obtains three results files:

- a geometric file
- a result file
- a chrono file with several time steps

Generally, the results are constituted of the following variables :

- the temperature
- the flux by band

Remark:
In SYRTHES *the radiation quantities are discretized in P0. It means that the quantities are constant by face.*

4.1.2.c Other files

These files which are not directly destined to be exploited by users, but in which it turns out to be useful to store information costly to generate. Typically, in the initial phase, it is necessary to calculate first the quantities necessary for the diverse interpolation between the conduction and the radiation grids. These quantities will be used in the resolution phase done at each time step.

Generating the view factors is generally regarded as being costly when the number of faces is high. One has to remember that the total number of view factors is equal to $n(n-1)/2$ if n is the number of independent faces.

SYRTHES has an option which allows to store the necessary quantities on a file. Using this option prevent to recalculate the same quantities for the following calculations. Indeed, these quantities are purely geometrical ones and do not change if the geometry does not change.

Similarly, it is possible to save to file data on interpolations between meshes conduction and radiation. we can then escape to recompute these data.

4.1.3 Coupling SYRTHES with a CFD code

In the case, of the coupling with a CFD code, the file organization stays unchanged. The files relative to the fluid code are simply added to the initial SYRTHES files (see figure 4.2). In that case, one has first the usual SYRTHES files and secondly the result files relative to the fluid code.

- three optional results file can however be generated by SYRTHES. They allow users to visualize some results on the fluid skin coupled to the solid. The first file contains mesh of the fluid skin in contact with the solid. The second and third files contains the fluid temperature and the heat exchange coefficients provided by the CFD code

for the flux calculation. As usual the only difference between the second and third files is that these information are given for the last time step calculated for the former and for several time steps asked by the users for the latter.

- a unique listing file is provided, in which fluid and solid information are mixed together at each time step. Such a choice has been retained by the SYRTHES conceptors because the splitting in two different listing files would only bring confusion. However, the option of different printing level for the solid part and the fluid part has been maintained.

Remark:
In all cases, the solid result file can be used to do a sequel calculation

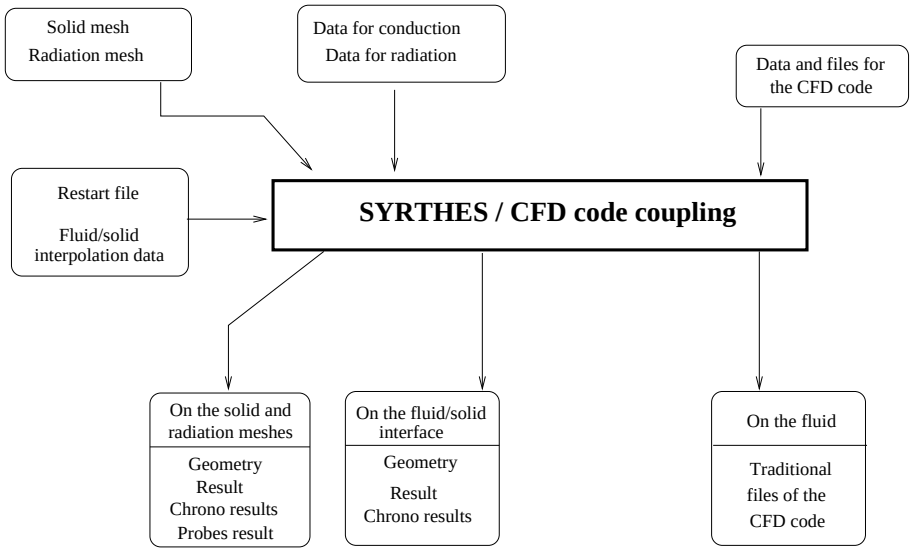

Figure 4.2: Global organization

4.2 Pre and post processors

As for any industrial calculations, the large quantity of data to input and generated makes absolutely compulsory the use of efficient pre and post processors. The format of fluid files being totally unaffected by a coupling with SYRTHES, the usual fluid post-processors stay available.

One prefers here to focus on the tools available for SYRTHES both for generating the necessary input data and analyzing the results obtained.

4.2.1 SYRTHES

Pre-processors
The unstructured mesh can be generated by any mesher. Users have only to make sure that the information generated are compatible with the ones required by SYRTHES.

For the time being, SYRTHES does recognize automatically files issued from SIMAIL (version 6.0) [14], and IDEAS-MS[15]. It is important to note, that SYRTHES accepts also files having the format SYRTHES (compulsory for the moving solids). This allows to do sequel calculations starting with a geometrical file coming from a previous calculation.

In practice, most people having a mesher not automatically recognized tend to generate the file directly to the SYRTHES format. Indeed, this geometric file contains all (and only) the necessary information, moreover, the format is easy to understand.

Post-processors
Whatever the option retained, SYRTHES always provides the file containing the geometry at the SYRTHES format (nodes coordinates + connectivity + references). Utility programs allow to transform these files in format compatible with post-processors generally available. These utility programs are described in paragraph 4.2.2. Among, others, format for post-processors like ENSIGHT [16], RUBENS [17] or IDEAS-MS [15] are available.

The format of theses result files are given in appendix B.

4.2.2 Utility programs available

One presents here different utility programs developed by the two authors to facilitate the post-processing task with different post-processors.

syrthes2ensight : translation of a SYRTHES file in a data base directly understood by ENSIGHT "case". (handling of 2D and 3D cases).

usage : *syrthes2ensight syrthes_geom syrthes_resu fich_ensight*

- ▷ *syrthes_geom* : name of the geometrical file issued from SYRTHES,
- ▷ *syrthes_resu* : name of the result file issued from SYRTHES. This file can be either the result file containing the last time step, or the chrono file containing several time steps. In that case, the n time steps will be handled.
- ▷ *fich_ensight* : Name of the file, one wishes to give to the ENSIGHT format file.

syrthes2med : translation of a SYRTHES file in a data base MED [18] [19].
usage : *syrthes2med syrthes_geom syrthes_resu fich_med*

▷ *syrthes_geom* : name of the geometrical file issued from SYRTHES,

▷ *syrthes_resu* : name of the result file issued from SYRTHES.

▷ *fich_med* : Name of the file, containing the mesh and result with a format MED.

The data files relative to SYRTHES 5

To run, the code SYRTHES requires at least a parameter file where the user can define his problem, his numerical choices, a geometrical file for the description of his solid domain, and finally an organization file where all the necessary paths to the diverse files are indicated. In case the radiation functionalities are activated, complementary files like the radiation mesh or the radiation parameter file are added.

5.1 Geometry files

5.1.1 Conduction grid

Let's remind first that this file is compulsory. This file contains the mesh as well as the references on the nodes, the elements, edges or faces.

The solid mesh being unstructured, the information necessary to SYRTHES are the coordinates of the nodes, the references of the nodes (possibly the edges and or faces) and the connectivity. **The elements need imperatively middle nodes**. One reminds that the only conduction elements treated by SYRTHES are 6 nodes triangles in 2D and 10 nodes tetrahedra in 3D). Moreover the edges are necessarily straight lines or the faces planar.

5.1.2 Radiation grid

In case the radiation functionality is activated, a second grid has to be provided to SYRTHES.
This mesh is also unstructured and contains the coordinates of the nodes as well as the connectivity, the references on the segment or the faces. This time the element are of the P1 type, i.e they do not have middle nodes. One reminds that the only element accepted by SYRTHES are 2 nodes segments in 2d and 3 nodes triangles in 3D.

5.1.3 Formats of the geometrical files

In its standard version, SYRTHES accepts three different formats :

- the files issued from SIMAIL (version 6.0),

- the files issued from SIMAIL IDEAS-MS,
- the file at the format SYRTHES issued from the code itself.

The identification of the format of the file is done automatically thanks to the extension given to the geometrical file. Consequently, this extension is fixed and compulsory :

- format SIMAIL $\longrightarrow$ "**.des**" (example : *toto.des)*,
- format IDEAS-MS $\longrightarrow$ "**.unv**" (example : *toto.unv)*,
- format SYRTHES $\longrightarrow$ "**.syr**" (example : *toto.syr)*,

It is advised to consult references [14] and [15] for the description of the SIMAIL and IDEAS-MS format. Appendix B describes the SYRTHES format of the geometrical file.

5.2 Parameters file

These two files contain all the user parameters to describe a problem. The following chapter is entirely devoted to these files.

5.3 Organization file : "syrthes.env"

It is the **only** file whose name is **totally fixed**. Its other particularity is that it is the only one which has to be located in the same directory as the executable.

Its aim is absolutely essential : indicate to SYRTHES all the paths to the files necessary during the run. This file can also help users to organize the work, since it allows users to give the most appropriate names for the different files as well as their locations.

5.4 Activation of the fluid/solid coupling

In the case of fluid/solid coupled calculation, SYRTHES is activated thanks to a keyword of the fluid parameter files (see the fluid code *Code_Saturne*user manual [2]).

It is important to underline, that for users, the activation of this keywords constitutes the only change compared with a standard use of the CFD code, since all the aspects relative to SYRTHES are handled independently in SYRTHES files.

For the solid part, the user will define the parameters in datafiles of SYRTHES as in the case of non-coupled calculations. Simply, some additional parameters appear in the data files.

Parameters files 6

Let's underline first that the names of the files noted here "*syrthes.data*" and "*syrthes.ray*" are not fixed. Users may wish to change them. However, it may be convenient to keep these names (for hot line purposes for example). The first file ("*syrthes.data*") contains all the parameters necessary for conduction, and the second one ("*syrthes.ray*") is specific to the transparent radiative aspects.

These files are constituted of keywords, most of them have default values.

6.1 Input file for conduction: *syrthes.data*

Even if the keywords order is free, we preserve generally the default value Proposed, in order to keep the file a certain logic which is proposed by failing to file the concerver a certain logic.
The file propose 3 major sections :

- **keywords to managing the calculation**
  We defines the main options from the calculation: time step, options solver, results management code, etc.
- **The definition of the references of nodes and faces of the mesh**
  In this paragraph, a link is established betwen the mesh entities and physical conditions to which they correspond.
- **The boundary conditions and physical properties**
  Are defines physical properties materials, initial conditions, boundary conditions,

Most parameters have default values and in practice users would have to change only a limited number of them.

6.2 Generalities

```
/ Definitions
/-------------
/
'TITRE POUR LE CALCUL SOLIDE=' 'CALCUL SYRTHES : test'
/
'DIMENSION DU PROBLEME=' 2
'AXE D AXISYMETRIE (AUCUN,OX,OY)=' 'AUCUN'
'SUITE DE CALCUL=' 'NON'
/
'ISOTROPIE DU MATERIAU='   1
'NOMBRE DE DIRECTIONS PERIODIQUES='   0
/
'CONDITIONS LIMITES PAR NOEUD OU PAR FACE='  'FACE'
'DEFINITION DES PROPRIETES PHYSIQUES PAR=' 'ELEMENT'
'DEFINITION DES FLUX VOLUMIQUES PAR=' 'ELEMENT'
```

6.2.1 Title

Keyword : TITRE POUR LE CALCUL SOLIDE

Here, it is possible to give a title to the current study.

6.2.2 Problem dimension

Keyword : DIMENSION DU PROBLEME

This keyword has two possible values :

2 : Calculation in 2D

3 : Calculation in 3D

6.2.3 Axisymmetric axis

Keyword : AXE D AXISYMETRIE

'AUCUN' : The calculation is considered to be cartesian

'OX' : Axisymmetric calculation where the prescribed axis is Ox

'OY' : Axisymmetric calculation where the prescribed axis is Oy

Keyword : SUITE DE CALCUL

This keyword indicates if it is a first calculation starting from time step 1, or if it is a sequel calculation.

'OUI' : If it is a sequel calculation. In that case the result file of the previous calculation has to be provided as starting file.

'NON' : this is not a sequel calculation. The computation will start right from the beginning (time step 1).

Keyword : ISOTROPIE DU MATERIAU

This keyword is an integer and can take different values :

1 : Isotropic material indicates that all materials have an isotropic behavior : the thermal conductivity is defined by a scalar (possibly variable with time and space) on each node of the domain,

2 : Orthotropic material means that at least one material has an orthotropic behavior : the thermal conductivity is defined by a diagonal matrix (whose terms are possibly function of time and space) on each node of the domain. In that case the isotropic material of this problem will be defined by setting all the component of the diagonal are equal.

3 : Anisotropic material means that at least one of the material behaves anisotropically : the conductivity is then defined by a symmetrical matrix 3×3 in 3D, and 2×2 in 2D. The components of this matrix may be function of time or space. Isotropic materials are defined by setting equal terms for the matrix, and 0 in extra diagonal terms.

Keyword : NOMBRE DE DIRECTIONS PERIODIQUES

It is possible to define frequencies up to 2 in dimension 2 and up to 3 in dimension 3. If the keyword is 0, no periodicity in set throughout domain.

6.2.4 Boundary conditions

Keyword : CONDITIONS LIMITES PAR NOEUD OU PAR FACE

This keyword indicates if SYRTHES must take into account the boundary conditions by node or by face.

'FACE' : in that case, the boundary conditions (apart the Dirichlet condition) are imposed on the boundary **faces** of the mesh. This option allows a proper handling of discontinuous boundary conditions. Indeed, a node can be seen differently according the element from which it is considered. If possible, this option should be preferred.

'NOEUD' : in that case, the boundary conditions are treated by **nodes**. This option still stays interesting for mesh generator which do not offer face references in standard. Indeed, generating the face references in a user subroutine, even if it is possible in SYRTHES may appear a little bit tricky for complex 3D problems. Moreover, in most cases the treatment by nodes is still sufficient to ensure a good handling of the boundary conditions (see example 3.5)

It should be noted that a coupling (either with radiation or a CFD code) is treated as being a boundary condition from the solid point of view. Consequently the coupling is managed by the choice made here.
It is recommended to use the definition by FACE for the boundary conditions

Keyword : DEFINITION DES PROPRIETES PHYSIQUES PAR

According the mesh generator used and the kind of problem treated, the physical properties of the material (density, specific heat, thermal conductivity) can be set on :

'NOEUD' : in that case the physical properties are set on the **nodes** of the mesh. This option stays interesting in the case where the elements have not been referenced. But one has to admit that it is not natural. Moreover, this choice does not allow to take into account a discontinuity of behavior of two different materials in contact. Indeed, an ambiguity remains on what material should be affected to the interface nodes.

'ELEMENT' : in that case the physical properties are defined on the element. This is the most common option and well adapted to constant properties.

'NOEUD PAR ELEMENT' : in that case, the physical behavior can vary inside the element. With this option, the properties may be function of local quantities on each node of the current element. The possibility to handle the discontinuity from one element to another is kept.

Keyword : DEFINITION DES FLUX VOLUMIQUES PAR

The behaviour is very similar to that of material properties (definition des proprietess physique par).

6.3 Calculation management

```
/ Pas de temps
/---------------
'PAS DE TEMPS SOLIDE='   100.
/'PAS DE TEMPS AUTOMATIQUE='  variation_temper(C) dt_max(s)
/'PAS DE TEMPS AUTOMATIQUE='  0.1 1000.
/'PAS DE TEMPS MULTIPLES='   iteration_max   pas_de_temps
/'PAS DE TEMPS MULTIPLES='   100 1.
/'PAS DE TEMPS MULTIPLES='   200 10.
'NOMBRE DE PAS DE TEMPS SOLIDES=' 3
```

6.3.1 The time step

Keyword : PAS DE TEMPS SOLIDE

If the time step is constant, its value is given in second.

Keyword : PAS DE TEMPS AUTOMATIQUE

The time step is automatically recalculated by the code each iteration. The user provides a value of maximum variation of temperature between two consecutive time step and the code calculates the time step in order to meet this maximum variation. The user also provides a maximum value of time step.

Keyword : PAS DE TEMPS MULTIPLES

The user sets several time step during calculation. The time step is constant in blocks of n iterations. For each block, it provides the value of iteration number (absolute) Max of block and the value of time step (in seconds) to apply on this block.

Keyword : NOMBRE DE PAS DE TEMPS SOLIDES

This keyword defines the number of time step for simulation to be done. We notice that the number of time step is always indicated in absolute. If 10 time steps have be done during a first calculation, and you want to perform 10 others during a series of calculation, the value to be indicated is 20.

6.4 Correspondence management

This paragraph is to inform that in the case of calculations coupled fluid / solid.

```
/ Gestion des correspondants
/--------------------------
'STOCKAGE DES CORRESPONDANTS SUR FICHIER=' 'OUI'
'LECTURE DES CORRESPONDANTS SUR FICHIER='  'NON'
```

Finding the correspondence between the solid and fluid domains (allowing the transfer between the two domains) may prove to be CPU consuming, especially if the number of nodes located on the fluid/solid interface is large. Even, if such a task has to be performed only once, during the initialization phase, it is interesting to do this calculation only once per study instead of once per calculation. To do this, the information required are stored in a file the first time a calculation is performed. Then for the following calculations, or trials, all is needed is to read the correspondent file.

Keyword : STOCKAGE DES CORRESPONDANTS SUR FICHIER

- **Oui** : The correspondents are stored in the file
- **Non** : The correspondents are not stored in the file

Keyword : LECTURE DES CORRESPONDANTS SUR FICHIER

- **Oui** : The correspondents are simply re-read on file
- **Non** : The correspondents are calculated

Remark 1 :
The correspondents can not be read and stored at the same time, however, they can be neither read nor stored.
Remark 2 :
The purpose of this file is limited to an internal SYRTHES *use. Any modification of this file could bring nasty consequences. Likewise, if some changes have occurred in either the fluid or the solid mesh, it is compulsory to recalculate the correspondents.*
Voluntarily, this file has been written in a formatted way (ASCII coding). The only purpose is to make this file computer independent. One can for example imagine to calculate the correspondent file on a CRAY computer, and use it later on a workstation or inversely.

6.5 Management of the output

```
/ Sorties
/--------
'NIVEAU DES IMPRESSIONS POUR LE SOLIDE=' 2
'ECRITURE MAILLAGE SOLIDE='     'OUI'
'PAS DES SORTIES CHRONO SOLIDE='    -1
/
'HISTORIQUES CONDUCTION='             'NON'
'CHAMP DE TEMPERATURES MAXIMALES='   'NON'
/
'ECRITURE MAILLAGE PEAU FLUIDE='     'NON'
'ECRITURE RESULTATS PEAU FLUIDE='    'NON'
'ECRITURE CHRONO PEAU FLUIDE='       'NON'
```

Keyword : NIVEAU DES IMPRESSIONS POUR LE SOLIDE

This keyword allows the user to manage the level of printing in the listing file, (in a completely decoupled manner from the fluid if a conjugate heat trasnsfer case). This turns out to be particularly flexible when results on one or the other domain is not completely satisfying.

This printing level is governing only the solid part. It can take the following values :

0 : no information is provided at all, this is not recommended especially for a first calculation.

1 : only the crucial information are included in the listing especially during the initial phases. At each time step, the solver convergence is indicated.

2 : the printing level is normal, which means that all information necessary to understand the case are printed, especially in the initial phase, The iteration of the solver are also given.

3 : This is the most detailed level. All information necessary to users are included in the initial phase, and the solver information is similar to the previous level.

Keyword : ECRITURE MAILLAGE SOLIDE

Keyword allows the management of a mesh file at the Syrthus format. This option may be interesting in the aftermath of calculations if the mesh is very big. During the initial calculation, file geometry result has been generated and it is then possible to save time not writing it any more. beware, however, that it is essential to generate this file at least once in ordre to generate post processing files in order to (For example EnSight).

oui : writing file geometry result

non : no writing.

Keyword : PAS DES SORTIES CHRONO

This keyword allows to manage the output of temperature field at intermediate time steps in the file "chrono". It is therefore possible to store the complete temperature field (values of the temperature on each node) at different time step. It should be noted however that such a possibility is to be used with some care when confronted to very large meshes. Indeed, storing too many time steps leads quickly to very big files.

-1 : the "chrono" file is not generated

n : the temperature field is all saved every n time step. Then the user has to enter the value of n (n is a positive integer).

6.5.1 Solid probes results

In the case of very large meshes, it may be sometimes difficult (for limited storage reasons), to save lots of time steps. It is therefore interesting to have an option allowing to follow the temperature evolution of a limited number of nodes. It is then possible to monitor some strategic locations, or simply facilitate the convergence appreciation.

Keyword : HISTORIQUE SOLIDE

oui : writing probes file

non : no writing.

If solid probes are wanted, one has to provide :

- the frequency at which the temperature of the solid probes should be saved in a probes result file. The frequency has to be given in seconds.
- the list of the nodes corresponding to probes. The format is free, and can be given on several lines.

6.5.2 Field of maximum/minimum temperatures

During a calculation, it may be intersting to know the maximum/minimum temperature reached at each node. The corresponding fields are not physical but representative of the maximum temperature that has been achieved during the transition for each node.
Keyword : CHAMP DE TEMPERATURES MAXIMALES

oui : calculating and writing the field of maximum temperatures,

non : no calculating.

6.5.3 Management of the output on the fluid skin

In some fluid/solid coupled problems, it can be interesting to know quantities exchanged at the interface. The quantities written by SYRTHES are the local near wall fluid temperature and the heat exchange coefficient. These results are given under the same format as the solid result. They appear under the form of three files :

- a geometrical file of the fluid skin,
- a result file corresponding to the last time step,
- a chrono file (intermediate results).

In order to keep the structure of the code as simple as possible, the frequency of this skin "chrono" file is identical to the solid "chrono" file. Consequently, it is necessary to define the solid "chrono" first. The skin output are managed thanks to three keywords :

Keyword : ECRITURE MAILLAGE PEAU FLUIDE

Keyword : ECRITURE RESULTATS PEAU FLUIDE

Keyword : ECRITURE CHRONO PEAU FLUIDE

6.6 Numerical choices

```
/ Choix numeriques
/-----------------
'NOMBRE ITERATIONS SOLVEUR SOLIDE='   100
'PRECISION POUR LE SOLVEUR SOLIDE=' 1.E-6
```

The thermal equation in the solid is solved thanks to an iterative preconditioned conjugate gradient method. The precision required can be set by users. Two keywords are offered to control the precision required and the CPU time required. In practice, it is tradeoff between the precision and the CPU cost.
Keyword : NOMBRE ITERATIONS SOLVEUR SOLIDE

Keyword : NOMBRE ITERATIONS SOLVEUR SOLIDE (relative convergence)

It is important here to precise one technical point: how the code stops the iterative procedure. This stop is based on three criteria :

- the maximum number of iteration ,
- an absolute convergence criterium,
- a relative convergence criterium.

The algorithm stops therefore either when :

- the maximum number of iterations allowed has been reached,
- both convergence criteria are respected.

Thus, users can control completely the solver :

- ask for a very high precision: by imposing a very severe relative convergence criterium and indicating a maximum number of iteration sufficiently large.
- ask for an intermediate precision and/or limit the maximum number of iterations allowed.

Finally let's precise the definition of the convergence criteria implemented in SYRTHES.

$$\begin{aligned} absolute \qquad & \|Ax - b\| < \varepsilon_1 \\ relative \qquad & \frac{\|Ax - b\|}{\|x_n\|} < \varepsilon_2 \end{aligned}$$

where ε_1 is fixed (set to $\varepsilon_1 = 10^{-4}$), ε_2 given by users and where x_n is the result of the previous time steps.

6.7 Definitions of references

```
/ Choix numeriques
/-----------------
'REFERENCES NOEUDS OU FACES SOLIDES COUPLE(E)S' 0
'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   0
'REFERENCES NOEUDS OU FACES SOLIDES AVEC FLUX' 0
'REFERENCES NOEUDS OU FACES SOLIDES AVEC COEFFICIENT D ECHANGE' 0
'REFERENCES NOEUDS OU ELEMENTS SOLIDES AVEC FLUX VOLUMIQUES' 0
'REFERENCES NOEUDS OU FACES SOLIDES AVEC RESISTANCE DE CONTACT' 0
'REFERENCES NOEUDS SOLIDES PERIODIQUES' 0
'REFERENCES NOEUDS OU FACES SOLIDES AVEC RAYONNEMENT INFINI'  0
```

Are indicates here the correspondence between references imposed on the boundary conditions and the kind of boundary conditions. For each item,a list of references corresponding to a given type. .List reduced to " 0" means that condition of this type is desired

Remark : *the fact that we consider references eihter or nodes or faces is managed by the keyword previously :* CONDITIONS LIMITES PAR NOEUD OU PAR FACE

Example :

```
'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   1 23  5  3
'REFERENCES NOEUDS OU FACES SOLIDES AVEC FLUX' 12  4  6   10
```

6.8 Initial conditions

```
/ -------------------------------
/ Entree des conditions initiales
/ -------------------------------
/ mot-cle       valeur       liste des references
'CINI'           25.               1 4  2 12
'CINI'           30.               5 7
```

The default value is to set a uniform temperature of 20°C.

- $T_{initial}$: initial temperature in degrees Celsius,
- *References* : list of the references of the nodes subjected to this initial temperature (a value set to -1 indicates that **all** solid nodes are considered).

6.9 Boundary conditions

```
/ -------------------------------------------------------
/ Entree des conditions aux limites constantes par bloc
/ -------------------------------------------------------
/ mot-cle  type           valeur       liste des references
/
/'CLIM'    'FLUX'           flux.              1
/'CLIM'    'DIRICHLET'      T                  2
/'CLIM'    'COEF ECH'       Text   h           3
/'CLIM'    'COEF ECH'         20 6.3         2
/'CLIM'    'RES CONTACT'     g                 4
/'CLIM'    'PERIODICITE' 'T' vx vy vz       1 3 -1 4 5
/'CLIM'    'PERIODICITE' 'R' vx vy vz a1 a2 a3 b1 b2 b3 1 2 -1 3 4
/'CLIM'    'RAYT INFINI'     T  emiss          5
/'CLIM'    'RAYT INFINI'     20.   1.       2
```

The following boundary conditions are possible :

- **Flux** in W/m^2 (default value : 0 W/m^2)
- **Dirichlet** : Temperature in °C. (default value : 20°C)
- **Exchange** : The heat exchange coefficient is given in W/m^2K and the external temperature in °C. (default value : T=20°C, h=0 W/m^2K)
- **Contact resistance** : Resistance in W/m^2K. (default value : 0 W/m^2K, meaning a total cut)
- **Periodicity** This option is maybe less straightforward than the others, and will therefore be explained in detail in the following paragraph.

- **Infinite radiation**.

Remark1 :
For all these keywords, a list of references reduced to -1 *means that* **all** *nodes or faces are subjected to this condition.*
Remark2 :
All boundary for which no condition has been specified will implicitely carry a zero flux (or adiabatic condition)

How to define periodicity

It is also possible to read paragraph 3.1.5 to get further information on what is really the periodicity option.

It is possible to define two kinds of periodicity :

- **translation**
  In that case, users have to provide a translation vector (V_x, V_y, V_z) which allow to go from the first boundary towards the second boundary. It should be noted that first or second boundary is simply a convention matter, but once the choice made, the translation vector has to be coherent with the choice made. Thus in the following example, if the right side which has been chosen as being the "boundary 1". Consequently the translation vector is negative. It is during the setting of the periodic references lists that the two kind of boundaries are implicitely defined.

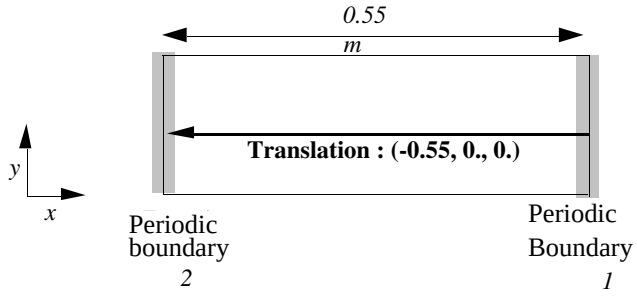

Figure 6.1: Definition of the translation vector

- **rotation**
  In that case, users have to give :

  - the translation vector between the global system of coordinate and the local system of coordinate,
  - the three angles of rotation, expressed in degrees respectively around the axis x, y and z) transforming the global system of coordinates in the local system of coordinates.
  - the 3 angles (expressed in degres) describing the rotation of the body itself (respectively around the local axis x, y et z),
  - the references identifying the boundary 1 and the references identifying the boundary 2.

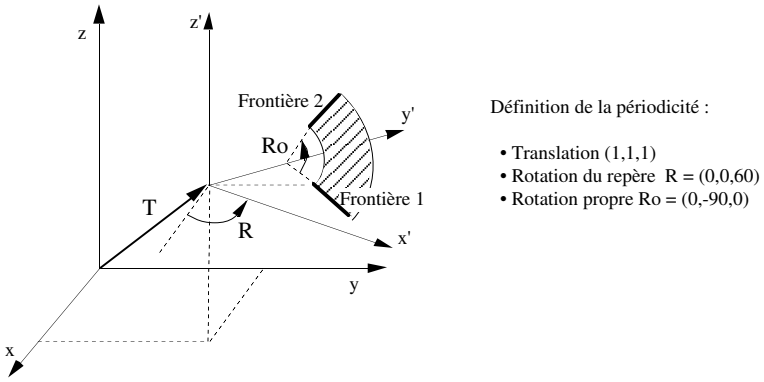

Figure 6.2: Definition of a rotation

In short, we must never forget that the transformation Geometric which is defined must describe the transformation that lets you go from 1st references group to the second.

6.10 Volumetric flux

```
/'CVOL'       value     references
'CVOL'      1200          13  17
```

The default value is 0(no source term)

- *value* : value of volumetric flux in W/m^3
- *references* : list of references of the nodes or element injected to this source term. A value of -1 means that all entities are set to the given source term. (see paragraph (6.7)for the list declaration)

The condition "flux volumique" can be applied in nodes or element according the value given to the keyword "definition des flux volumique par"(cf 6.2.4).
A list reduced to -1 *mens that* **all** *nodes or elements are subjected to the given value*

6.11 Material properties

```
/ mot-cle   type           valeur       liste des references
 'CPHY'     'RHO'           7700.       -1
 'CPHY'     'CP'            460.        -1
 'CPHY'     'K ISOTROPE'    25.        -1
/
/ 'CPHY'  'K ORTHOTROPE'      k11 k22 k33              ref
/ 'CPHY'  'K ANISOTROPE'      k11 k22 k33  a1 a2 a3    ref
```

The code requires absolutely the following quantities :

- ρ : density (default value 7700 kg/m^3),
- k : thermal conductivity (default value 25.1 W/mK),
- C_p : specific heat (default value 460 $J/kg\,K$).

According the characteristics of the solids, the following informations have to be given :

- isotropic material
  - ▷ density (kg/m^3) : 'CPHY' 'RHO'
  - ▷ thermal conductivity (W/mK) : 'CPHY' 'CP'
  - ▷ specific heat ($J/kg\,K$): 'CPHY' 'K ISOTROPE'
- orthotropic material
  - ▷ density (kg/m^3) : 'CPHY' 'RHO'
  - ▷ thermal conductivity ($J/kg\,K$) : 'CPHY' 'K ORTHOTROPE'
  - ▷ specific heat (W/mK) : 'CPHY' 'CP'
    One has to give 2 or 3 values (according the dimension of space); the conductivity along x, the conductivity along y, and eventually along z
- anisotropic material
  - ▷ density (kg/m^3) : 'CPHY' 'RHO'
  - ▷ specific heat (W/mK) :'CPHY' 'CP'
  - ▷ thermal conductivity ($J/kg\,K$) : 'CPHY' 'K ANISOTROPE' in 2D one has to give two values for the conductivity in the local coordinate system relative to the body, then the angle of rotation of the coordinate system around the axis z. In 3D, one has to specify 6 values, first the conductivity along the axis x, y and z of the local system of coordinates, and then the 3 angles (in degrees) corresponding to the rotation around the axis x, y and z describing the rotation (from the global system of coordinates towards the local system of coordinates).
    Remark : *in 2-dimensional, only conductivity in x and y and z angle around will be non-zero*

Figure 6.3 presents an example to define an anisotropic conductivity in the case of disk. The ellipses indicates the propagation of the heat function of the conductivity. In both cases the conductivity is 25 W/mK in the first direction and 5 W/mK in the second one. If the local system of coordinate is aligned with the global system in the left example, one has a 45° angle around z in the right example.

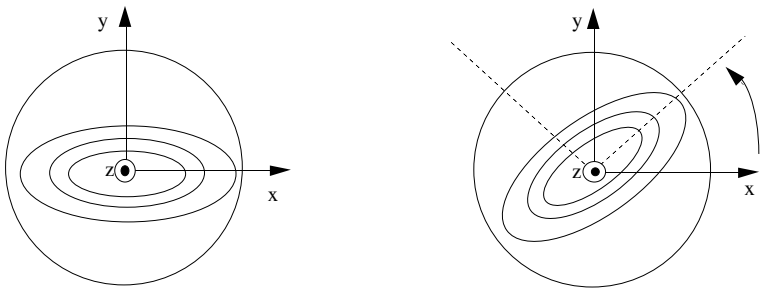

Figure 6.3: Anisotropic conductivity

```
/ 'CPHY'  'K ANISOTROPE'      k11 k22 k33  a1 a2 a3    ref
'CPHY'  'K ANISOTROPE'          25. 5. 0.   0. 0. 0.    1
'CPHY'  'K ANISOTROPE'          25. 5. 0.   0. 0. 45.   2
```

6.11.1 Historique

```
'HIST' 'FREQ' 100.
'HIST' 'NOEUDS' 1 34 143 4
'HIST' 'NOEUDS'  156 2203  12220
```

Keyword : 'HIST' 'FREQ'

It is here to provide the frequency of writing history. It is a value in seconds.

Keyword : 'HIST' 'NOEUDS'

This keyword allows you to provide a list of nodes on which wishes to have histories. This keyword may, if necessary, be invoked many times if the list of nodes is long.

6.11.2 Flux balance

```
'BILAN FLUX SURFACIQUES'  12 3
'BILAN FLUX VOLUMIQUES'  2 4
```

Keyword : BILAN FLUX SURFACIQUES Keyword : BILAN FLUX VOLUMIQUES

SYRTHES allows to calculate heat flux load balance either in boundary faces (BILAN FLUX SURFACIQUE), or either in volumetric element (BILAN FLUX VOLUMIQUE).
In both cases, users have to provide a reference liste and in return SYRTHESgive the flux in the corresponding faces. These keyword may appered several times in the data file. this allows to do heat flux balance in selected bondaries or volume.

Beware : this option is only available when bondary conditions are set in faces
In the listing file, SYRTHESprovide provider :

- The bondary heat flux : (induced by flux or exchange conditions)
- The convective flux :, in this case the face is coupled with CFD code (conjugate heat transfer simulation).
- The radiation flux : in this case the face is coupled with confined radiation.

Regarding the volumitric flux, it is related to the energy balance mading by the volumetric source terme. The result is given in W.

6.12 Input file for radiation : *syrthes.ray*

The file is composed of 3 " major"paragraphs" :

- **keywords for managing radiation**
  We define the main options calculation: periodicity, view factors, spectral bands,...
- **The definition of reference faces mesh**
  In this paragraphs we relate references of the mesh and physical conditions.

- **The data physical conditions, boundary conditions ,...**
  We defines the physical properties materials, the boundary conditions, ...

Most parameters have default values and in practice user has to change only a limited number of them.

6.13 Radiation

6.13.1 Definition

```
/ Definitions
/ -----------
'PERIODICITE DE ROTATION POUR LE RAYONNEMENT='  'NON'
'NOMBRE DE PLANS DE SYMETRIE POUR LE RAYONNEMENT=' 0
/
'NOMBRE DE BANDES SPECTRALES POUR LE RAYONNEMENT=' 1
```

Keyword : PERIODICITE DE ROTATION POUR LE RAYONNEMENT
Only rotation periodicity can be handled due to the closed domain constraint.

Oui : periodic condition exist.
Non : The domain has no periodicity.
Becareful:
It is compulsory to have the same node distribution on the boundaries.

Keyword : NOMBRE DE PLANS DE SYMETRIE POUR LE RAYONNEMENT
When the geometry has symmetries, it is possible to perform calculations only on the reduced portion of the domain (1/2, 1/4 or even 1/8 in dimension 3). Users have then to indicate the number of symmetries present in their problem.

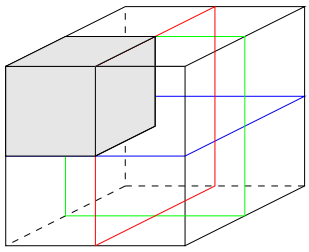

Figure 6.4: Example of symmetry in dimension 3

Becareful:
Specifying symmetries is not possible if they do not lead to a closed domain (as in the following example).

Typically, the situation where two symmetry planes are facing each other (to model are infinitely long) is not handled by SYRTHES.

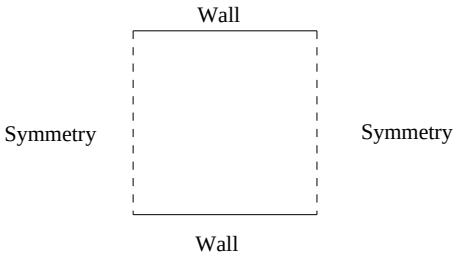

Figure 6.5: Example of a non authorized situation

The number of symmetry planes can take the values 0 to 2 in dimension 2, and from 0 to 3 in dimension 3.
The default value is 0.
Keyword : NOMBRE DE BANDES SPECTRALES POUR LE RAYONNEMENT
It is possible to take into account the spectral aspect of the thermal radiation. The bodies are then considered as gray by band. The emissivity by band has then to be provided.

The maximum number of bands handled has been limited to 100 (which is already considerable and is likely never to be a limitation).

6.13.2 Définitions des sorties du code

```
/ Sorties
/ -------
'NIVEAU DES IMPRESSIONS POUR LE RAYONNEMENT=' 2
'PAS DES SORTIES CHRONO RAYONNEMENT='  -1
'HISTORIQUES RAYONNEMENT='NON'
```

Keyword : NIVEAU DES IMPRESSIONS POUR LE RAYONNEMENT

This keyword specifies how detailed is the listing regarding specifically the radiation aspects. indeed, One reminds that conduction and radiation parts are printed in the same output file. The following possibilities can be activated :

- **0** no printing at all
- **1** minimal printing level
- **2** normal printing level
- **3** detailed printing level

Usually, the "normal printing level" is recommended.

Keyword : PAS DES SORTIES CHRONO RAYONNEMENT
This keyword controls the writing frequency of the chrono result file specific to radiation. Possible options are :

- **-1** no output
- **n** every n time steps : user has then to specify the value n.

Keyword : HISTORIQUES RAYONNEMENT
Like for conduction, it is possible to record the temperature evolution of some faces. Users have to provide a list of faces (free format) of the retained faces. The temperature is then recorded at each time step.

OUI : Writing of the radiation probe file

NON : No files

6.13.3 Radiation correspondents and view factors management

```
/ Gestion des correspondants et facteurs de forme
/-----------------------------------------------
'NOMBRE DE REDECOUPAGES POUR CALCUL DES FACTEURS DE FORME=' 0
'DOMAINE DE RAYONNEMENT CONFINE OUVERT SUR L EXTERIEUR=' 'NON'

'STOCKAGE DES FACTEURS DE FORME SUR FICHIER=' 'OUI'
'LECTURE DES FACTEURS DE FORME SUR FICHIER='  'NON'
/
'STOCKAGE DES CORRESPONDANTS POUR RAYONNEMENT='  'OUI'
'LECTURE DES CORRESPONDANTS POUR RAYONNEMENT='  'NON'
```

Keyword : Nombre de redecoupages pour calcul des facteurs de forme In the case where the radiation mesh is not appropriately refined, it may be possible to improve the view factor accuracy by asking to split automatically patches when occlusion cases occur.

Becareful :
If this option may appear at first as very interesting, one has to be aware that it may lead to very long computation time. Moreover, it is allways preferable to build an appropriate mesh for the problem to be treated.
The possible values go from 0 to 2. It is recommended to start by setting the splitting level to 0.

Keyword : stockage des facteurs de forme sur fichier Keyword : lecture des facteurs de forme sur fichier
In some configurations, the view factors calculation may turn out to be costly. SYRTHES allows to calculate these purely geometrical quantities only once and then store these coefficients in a file. For restart calculations, this file may be read instead of recalculating the view factors. The possible values are :

- Calculation and writing on file
- Reading on file
- Calculation

Keyword : Gestion des correspondances pour le rayonnement

In some configuration, establishing the correspondents between the radiation and conduction meshes can turn out to be costly. SYRTHES allows to calculate these correspondents only once and then store the information in a file. For restart calculation, this file can be read instead of recalculating the correspondents. The possible values are :

- Calculation and writing on file
- Reading on file
- Calculation

6.13.4 Geometrical definitions

6.13.4.a Definition of the connex volumes

These data are used by SYRTHES to determine how to orientate the radiation mesh provided by users (i.e separate the interior from the exterior). Basically, it is necessary to provide as many internal points as the number of connex volumes appearing in the problem.

For each volume, the coordinates of an internal point have to be given. Even, if very complex algorithms try to detect a maximum of possible user errors, this keyword has to be set with caution. Indeed an error in the face orientation generally leads to very bad behaviour. One should be particularly carefull when defining the internal points.

Example :
One wishes to calculate the thermal field in a plate (see figure 6.6) having two holes in which radiation is taking place.

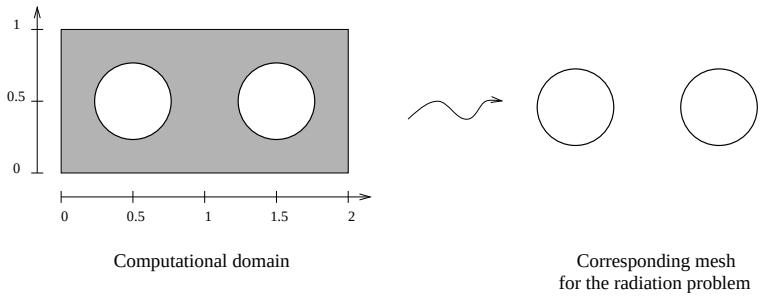

Figure 6.6: Typical domain of calculation including radiation

The domain has two holes (two connex volumes) which could be specified by the following manner :

```
/ 'RAYT'   'VOLUME CONNEXE'     Px   Py   Pz
'RAYT'   'VOLUME CONNEXE'      0.5   0.5  0.
'RAYT'   'VOLUME CONNEXE'      1.5   0.5  0.
```

6.13.4.b Symmetry plans

Definition
To define symmetry planes in a radiation mesh, users have to provide the plane equation coefficients (i.e $ax + by + c = 0$ in dimension 2 and $ax + by + cz + d = 0$ in dimension 3). These equations appear as many times as the symmetries appear in the problem.
Even if some test have been introduced in the code, some care has to be taken when using this option. Users should pay attention to the global geometry they want to simulate, and make sure that symmetries do effectively exist.

```
/'RAYT'   'SYM2D'  ax +  by +  c = 0
'RAYT'   'SYM2D'  0.    1.    0.
/
/'RAYT'   'SYM3D'  ax +  by +  cz + d = 0
/'RAYT'   'SYM3D'  0.    0.    1.  -0.5
```

6.13.4.c Periodicity

Definition of the periodicity
In dimension 2, one has to provide the coordinates P_x, P_y of the rotation axis, and the angle (in degrees) α between two periodic sections. In dimension 3, one has to provide the coordinates P_x, P_y, P_z of an invariant point, the axis of rotation component A_x, A_y, A_z and finally the angle (in degrees) α between two periodic sections.

```
/'RAYT'  'PERIO2D'   Px Py      alfa
'RAYT'  'PERIO2D'    0   0       90
/
/'RAYT'  'PERIO3D'   Px Py Pz    Ax Ay Az    alfa
/'RAYT'  'PERIO3D'   0  0  0       0  0  1     90
```

6.13.4.d Radiation domain open on the exterior

This keyword may be activated, when the radiation calculation is taking place in a non closed domain. This option is however to be used with caution, and closing the domain and setting an imposed temperature condition should generally be prefered.

```
'DOMAINE DE RAYONNEMENT CONFINE OUVERT SUR L EXTERIEUR=' 'NON'
```

6.13.5 Physical quantities

6.13.5.a Spectral bands

It is possible to take into account the spectral aspect of the thermal radiation. The bodies are then considered as gray by band. The emissivity by band has then to be provided. The maximum number of bands handled has been limited to 100 (which is already considerable and is likely never to be a limitation).
According the number of spectral bands indicated, the lower and upper bounds of each band has to be given. One reminds that spectarl bands limit have to be given in meters.

```
'NOMBRE DE BANDES SPECTRALES POUR LE RAYONNEMENT=' 1
/
/                              bande    lbd1     lbd2
'RAYT'  'BANDES SPECTRALES'     1     1.e-10      10.
/
/                               bande    emissi   ref
'RAYT'  'EMISSIVITE PAR BANDE'     1      0.95     -1
/
```

6.13.5.b Emissivity

Each material behaves differently regarding the radiation phenomenon. The quantity caracterizing this property is called the emissivity varying between 0 and 1 and is given by band as shown is the following example.

```
'NOMBRE DE BANDES SPECTRALES POUR LE RAYONNEMENT=' 2
/
/                               bande    lbd1     lbd2
'RAYT'  'BANDES SPECTRALES'      1     1.e-10       1.e-6
'RAYT'  'BANDES SPECTRALES'      2     1.e-6       10.
/
/                                 bande    emissi   ref
'RAYT'  'EMISSIVITE PAR BANDE'      1      0.95     1 2 3
'RAYT'  'EMISSIVITE PAR BANDE'      2      0.9      1 2 3
'RAYT'  'EMISSIVITE PAR BANDE'      1      0.45     4 5
'RAYT'  'EMISSIVITE PAR BANDE'      2      0.3      4 5
/
```

6.14 Radiation boundary conditions

The way to specify boundary conditions is fairly similar to what has been explained for the conduction case. The radiation mesh is divided in several parts identified through references (notion of color in the mesh generator IDEAS-MS). One reminds that these references are imposed on faces (subdomain notion in SIMAIL, or element color for IDEAS-MS). The following keywords allow SYRTHES to set the proper radiation boundary conditions. It is compulsory to fill these windows quite carrefully.

This section is devoted to the radiation faces not coupled to the solid.
The possible conditions are :

- Imposed temperature : the temperature of the face is imposed and given in $°C$.

  Becareful : *For coherence reasons, considering only such faces in the radiation domain is forbidden.*

  ```
  /'RAYT'  'TEMPERATURE IMPOSEE'   temp (degre C)        ref
  /'RAYT'  'TEMPERATURE IMPOSEE'      30.                  3
  ```

- Imposed flux by band. The flux is to be given in W/m^2.

  Becareful :

  For coherence reasons, considering only such faces in the radiation domain is forbidden.
  One may note that specifying such quantities, when grey bodies (by band) are considered, may turn out to be quite tricky.

  ```
  /'RAYT'  'FLUX IMPOSE PAR BANDE'    bande     flux (w/m2)    ref
  'RAYT'  'FLUX IMPOSE PAR BANDE'       1     1000       -1
  'RAYT'  'FLUX IMPOSE PAR BANDE'       2     1100       -1
  ```

6.15 Radiation input/output

6.15.1 Output management

```
'NIVEAU DES IMPRESSIONS POUR LE RAYONNEMENT=' 2
'PAS DES SORTIES CHRONO RAYONNEMENT='  -1
```

6.15.2 Radiation correspondents and view factors management

Keyword : Nombre de redecoupages pour calcul des facteurs de forme

In the case where the radiation mesh is not appropriately refined, it is possible to improve the view factor accuracy by asking to split automatically patches when occlusion cases occur.

Becareful :
If this option may appear at first as very interesting, one has to be aware that it may lead to very long computation time. Moreover, it is allways preferable to build a mesh appropriate to the problem to be treated.
The possible values go from 0 to 2. It is highly recommended to start by setting the splitting level to 0.

```
'NOMBRE DE REDECOUPAGES POUR CALCUL DES FACTEURS DE FORME=' 0
```

Keyword : Gestion des facteurs de forme In some configurations, the view factors calculation may

turn out to be costly. SYRTHES allows to calculate these purely geometrical quantities only once and then store these coefficients in a file. For restart calculations, this file may be read instead of recalculating the view factors.

```
'STOCKAGE DES FACTEURS DE FORME SUR FICHIER=' 'OUI'
'LECTURE DES FACTEURS DE FORME SUR FICHIER='  'NON'
```

Keyword : Gestion des correspondances pour le rayonnement

In some configuration, establishing the correspondents between the radiation and conduction meshes can turn out to be costly. SYRTHES allows to calculate these correspondents only once and then store the information in a file. For restart calculation, this file can be read instead of recalculating the correspondents.

```
'STOCKAGE DES CORRESPONDANTS POUR RAYONNEMENT='  'OUI'
'LECTURE DES CORRESPONDANTS POUR RAYONNEMENT='  'NON'
```

6.16 Coupling

The coupling between the different phenomena and meshes is done through references.

6.16.1 Conduction/radiation coupling

This paragraph has a meaning only if radiation from wall to wall is activated. It allows to define the parts of the radiation and conduction meshes which are coupled together.

Users have to set successively the list of references facing each other first on the solid conduction mesh and then on the radiation grid.

One reminds that on the radiation mesh, the notion of reference is always relative to faces, while for the conduction mesh, it can be either node or face according the option retained for the boundary conditions. (see the heading **"Boundary conditions"** where this choice is set by users).

```
/ References sur le solide
/ ------------------------
'REFERENCES NOEUDS OU FACES SOLIDES AVEC RAYONNEMENT CONFINE'  12 5 2
/
/ References sur le maillage de rayonnement
/ ----------------------------------------
'RAYONNEMENT : REFERENCES FACES COUPLEES AU SOLIDE'  4 7
```

6.16.2 Fluid/solid coupling

This paragraph has a meaning only if SYRTHES is coupled to a CFD code. It allows to define the parts of the fluid and solid meshes which are coupled together.

Users must successively set the list of references facing each other first on the solid conduction mesh and then on the fluid mesh.

One reminds that on the fluid mesh, the notion of reference is always relative to nodes, while for the conduction grid, it can be either node or face according the option retained for the boundary conditions. The user can see the notice User of *Code_Saturne*[2] for more details.

this keyword is to fill in the file *syrthes.data*

```
/ References sur le solide
/ ------------------------
'REFERENCES NOEUDS OU FACES SOLIDES COUPLE(E)S' 9 13 43
```

6.17 Files for the calculation

This heading allows users to indicate the file names (input data and output) to be used for the calculation. The window is splitted in 4 sections :

- SYRTHES **files organization**
  Users indicate here the different pathes to directories where the data file, the file for restart calculation and result files are located. users may choose to put all files in the directory or on the contrary to structure its study and classify these files.

  ```
  EMPLACEMENT DES FICHIERS POUR SYRTHES
    AMONT : ./
    SUITE : ./
    AVAL :  ./
  ```

  The path can be given relative to the position of the organization file (*syrthes.env*).

- **Names of input file**
  These files are required to define the calculation.

  ```
  NOM DES FICHIERS AMONT POUR SYRTHES
    DONNEES DU CALCUL : syrthes.data
    GEOMETRIE SOLIDE : maillage.des
    DONNEES POUR LE RAYONNEMENT :  syrthes.ray
    MAILLAGE RAYONNEMENT : maillray.des
  ```

- **Names of the file for restart calculation**
  Here the names of the file necessary to work out a sequel calculation are indicated.

```
NOM DES FICHIERS SUITE POUR SYRTHES
  SUITE SOLIDE RESU : resus1
  STOCKAGE DES CORRESPONDANTS :  corresp
  FACTEURS DE FORME RAYONNEMENT : fdf
  STOCKAGE DES CORRESPONDANTS RAYONNEMENT : corresp.ray
```

- **Names of the result files**
  Here the names of the different result files are indicated.

```
NOM DES FICHIERS AVAL POUR SYRTHES
  RESU SYRTHES 1 : geoms
  RESU SYRTHES 2 : resus1
  CHRONO SYRTHES 2 : resusc1
  HISTORIQUE SOLIDE RESULTAT : histos1
  MAILLAGE PEAU FLUIDE :  ff1
  RESULTATS PEAU FLUIDE : ff2
  CHRONO PEAU FLUIDE : ff2c
  MAILLAGE DE RAYONNEMENT : ray.geo1
  RESULTATS DE RAYONNEMENT : ray.res1
  CHRONO DE RAYONNEMENT : ray.chro1
  HISTORIQUE RAYONNEMENT : ray.histo1
```

According to the chosen options, the number of files needed or generated may differ.

6.17.1 SYRTHES calculation

6.17.1.a Input files

- Data file for SYRTHES
  This file is "" from an ASCII example. By convention, it is generally named *syrthes.data*.

- Solid mesh file
  This is the mesh describing the solid geometry. One underlines that the extention is compulsory and defines the type of mesh file (see 5.1).

6.17.1.b Restart files

- Restart file for the conduction
  When a restart calculation is done, one has to provide the result file of the previous calculation. Then the current calculation uses this file as a starting state.

6.17.1.c Results files

- Solid mesh
  This file contains the geometry of the case under the SYRTHES format. This file is used for post-pocessing purposes. It can also be used as a geometry input file. This file is compulsory when the moving solid option is activated. It contains then the updated geometry corresponding to the current state.

- Solid result file
  It contains the solid temperature at each node of the mesh.

- Solid chrono file
  It contains the solid temperature of each node of the conduction mesh every n time steps. This file does exist only if the chrono output have been activated.
- Thermal probes results file
  It contains the temperature value of specified nodes. This file does exist only if the option thermal probes has been activated.

6.17.2 Radiation calculation with SYRTHES

The files strictly related to SYRTHES stay unchanged. Only additional files are explicitely described here.

6.17.2.a Input file

- Input data file for the radiation
  This file is specific to the radiation phenomenon. It is generated through the graphical interface EDAMOX. By convention, is is named *syrthes.ray*.
- Radiation mesh
  It is a surfacic mesh discretizing the wall for the radiation calculation. One underlines that the file extension is compulsory and defines the kind of mesh file (see 5.1).

6.17.2.b Sequel files

- Radiation view factors file
  This file is only generated if users have specified the option reading or writing the view factors in a file. (see 6.15.2).
- Radiation/conduction correspondents file
  In the case where users have taken the option writing or reading of the correspondents, the name of the file is required (see (cf. 6.15.2).

6.17.2.c Results file

- Radiation geometrical file
  This file contains the radiation mesh. It is a surfacic mesh (segments in 2D and triangles in 3D). This mesh can be used for post-processing purposes.
- Radiation result file
  This file contains quantities relative to radiation (like flux, etc...) at the last time step calculated. Readers may refer to paragraph 4.1.2.b.
- Radiation chrono file
  This file is similar to the previous one, but contains results every n time steps.
- radiation probes.

6.17.3 SYRTHES calculation + coupling with a CFD code

The files purely relative to SYRTHES stay unchanged and are not discussed thereafter. Only additional files are presented.

6.17.3.a Input file

No additional file is necessary.

6.17.3.b Sequel file

- Fluid/solid correspondents file
  In the case where users have specified the reading or writing option, the name of the file is required (see **??**).

6.17.3.c Results file

- Fluid skin mesh file
  This file contains the mesh of the fluid skin identified as being in contact with the solid. It is therefore a surfacic mesh (segments in 2D, and triangles in 3D). This mesh is generated by SYRTHES for post-processing purposes, for example to present intermediate results on the fluid skin.
- Fluid skin results file
  This file contains information on all nodes of the fluid skin in contact with the solid. It is important to note that the quantities presented (local fluid temperature and heat exchange coefficient) are purely fluid ones directly transmitted from the CFD code without any transformation.
- Fluid skin chrono file
  This file is similar to the previous one except it contains the fluid quantities every n time steps on the fluid skin mesh.

It should be noted that these last files do exist only if users have activated the corresponding option.

The user subroutines 7

In order to make the use of SYRTHES easier, it is possible to define a certain number of conditions directly through the user-friendly graphical interface. Thus no programming is necessary and conditions and options can be altered with no need to recompile and link any program. If such an approach is very convenient from an ergonomy point of view, it may present some limitations. Typically, conditions may only vary from one part to another. It is impossible to taken into account transient conditions or behaviors. Likewise, temperature dependent material are not possible to handle.

A certain number of subroutines are therefore put at the users' disposal. Within these user-programs it is possible to set very complex conditions, depending of time, space, local temperature[1], etc...

A complete review of the user-subroutines is done in the following paragraphs. For each of them, their specificities are detailed. In each program, the beginning of the lines to be modified by users is indicated by the following message :

"USER INTERVENTION FROM THIS LINE"

It should be noted that each program is not entirely included in this document, but only the portion of particular interest for users. The comments relative to the programming lines appears in slanted characters.

7.1 Set up of the initial temperature : INITMP

This user-subroutine allows to set an initial temperature to the solid. It can be variable in space.

The variable to set is :

▷ $TMPS(i)$: temperature at the solid node i,

The figure 7.1 describes the initialization of the temperature.

7.2 Material properties : CPHYSO

The aim of this user-subroutine is to affect to each node, or element the material characteristics. The quantities to define are :

- density (kg/m^3)

[1] One reminds that the temperature in SYRTHES is always expressed in degrees Celsius.

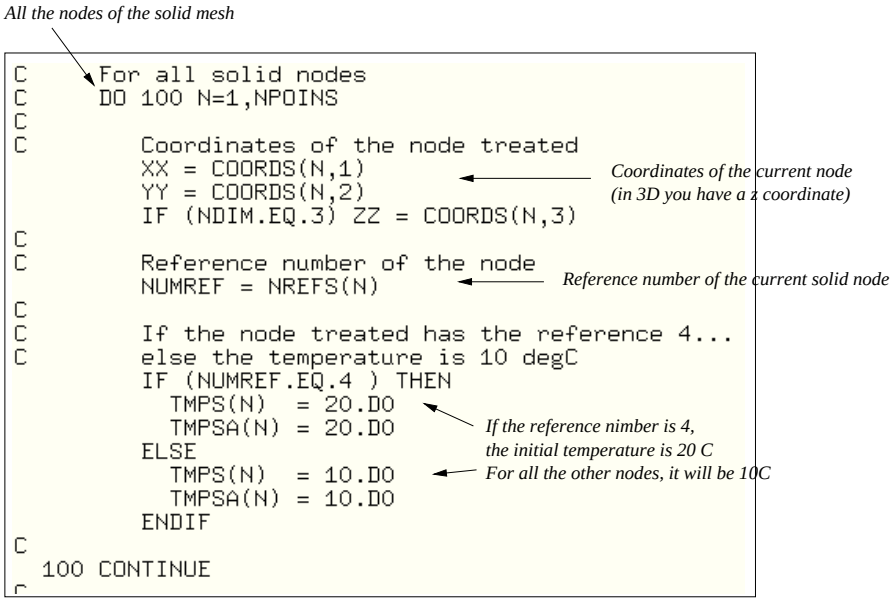

Figure 7.1: Programming initial temperature

- specific heat (J/kgK)
- thermal conductivity (W/mK)

The following figure presents the part of the user sub-routine *cphyso.F* dealing with the material properties at the **nodes** of the mesh. User can find in the same program similar sections but dealing with material properties defined on element or node by element.

According the choice done in the data file (see 6.2.4), users will fill out the corresponding section of the *cphyso.F* program.

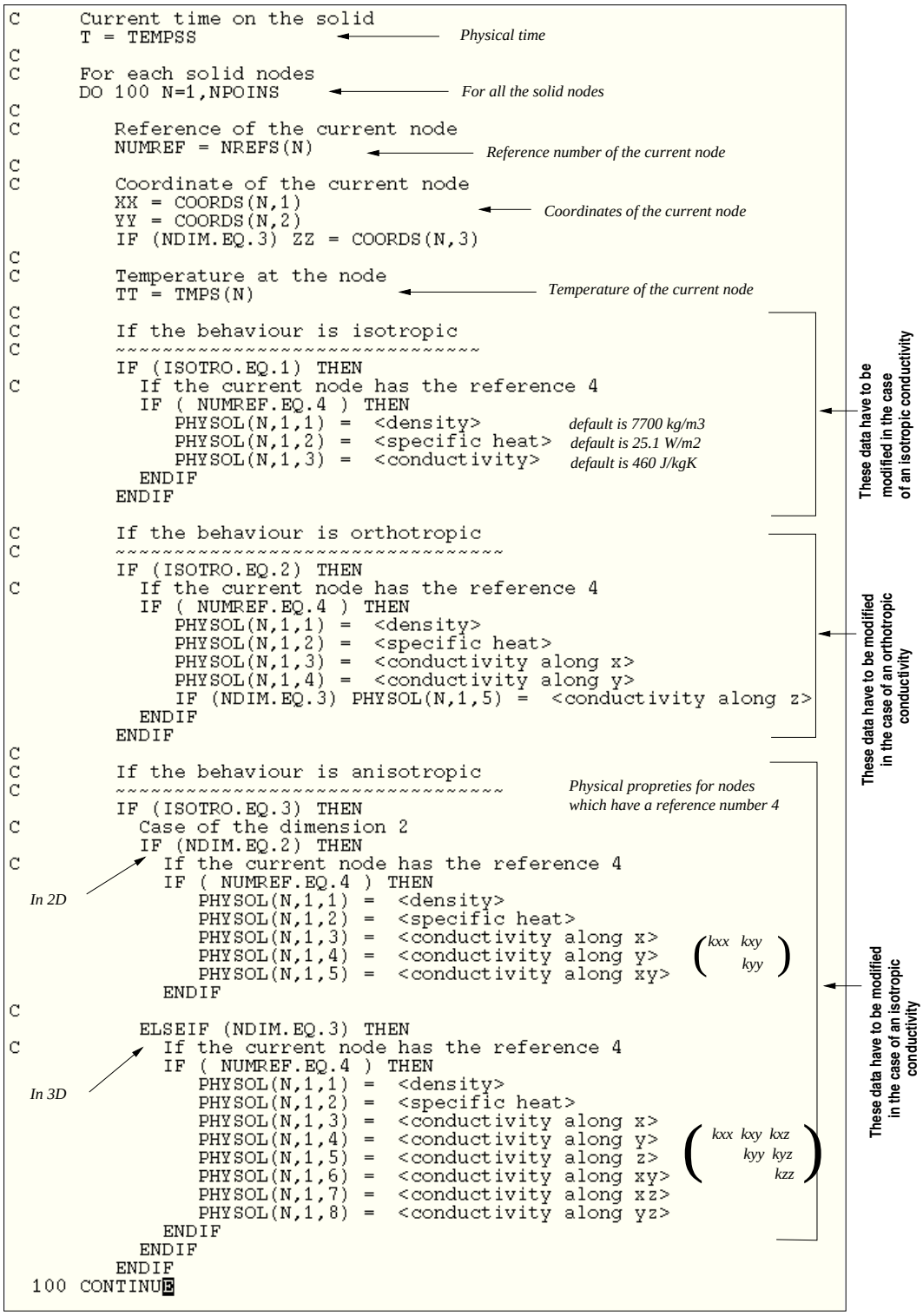

Figure 7.2: Programming material properties defined by node

7.3 Boundary conditions defined by node : LIMSOL

This user subroutine is to be used when the option 'Set up of the solid boundary conditions by' is set to 'Node'.

The part to program is splitted in several sections, each of them dealing with a different boundary condition. If in the problem treated, one of these conditions is not encountered, lines relative to this kind of condition have to stay commented.

```
C     ============================================================
C     2- INITIALIZATION OF THE BOUNDARY CONDITIONS OF THE KIND FLUX
C     ============================================================
C
C     Current time on the solid
      T = TEMPSS                          <-- Physical time
C
C     Example ...
C     For each node affected by a flux condition
      DO 200 N=1,NBFLUS                   <-- All the nodes affected by a flux boundary condition
C
C        Global number of the node
         NUMNO = NFLUSS(N)                <-- Global number of the current node
C
C        Reference of the node
         NUMREF = NREFS(NUMNO)            <-- Reference number of the current node
C
C        Coordinates of the node
         XX = COORDS(NUMNO,1)             <-- Coordinates of the current node
         YY = COORDS(NUMNO,2)
         IF(NDIM.EQ.3) ZZ = COORDS(NUMNO,3)
C
C        TT1 = Temperature(noted TT1 here) at the current node
         TT1 = TMPS(NUMNO)                <-- Temperature of the current node
C
C        If the current node has the reference 3, then the flux
C        id set to 1000 W/M2, otherwise  10000 W/M2
         IF (NUMREF.EQ.3) THEN            <-- For example, nodes can be distinguish by the reference number
            VFLUSS(N) = 1000.0D0
         ELSE
            VFLUSS(N) = 10000.0D0         <-- Value of the flux (W/m2)
         ENDIF
C
  200 CONTINUE
C
```

Figure 7.3: Programming of boundary conditions of the kind flux

```
C     ================================================================
C     3- INITIALIZATION OF THE BOUNDARY CONDITION OF THE KIND DIRICHLET
C     ================================================================
C
C     Current time on the solid
      T = TEMPSS                          <-- Physical time
C
C     Example ...
C     For each node affected by a dirichlet condition
      DO 300 N=1,NBDIRS                   <-- All the nodes affected by a Dirichlet boundary condition
C
C        Global number of the node
         NUMNO = NDIRS(N)                 <-- Global number of the current node
C
C        Reference of the node
         NUMREF = NREFS(NUMNO)            <-- Reference number of the current node
C
C        Coordinates of the node
         XX = COORDS(NUMNO,1)             <-- Coordinates of the current node
         YY = COORDS(NUMNO,2)
         IF(NDIM.EQ.3) ZZ = COORDS(NUMNO,3)   <-- In 3D only
C
C        TT1 = Temperature(noted TT1 here) at the current node
         TT1 = TMPS(NUMNO)                <-- Temperature of the current node
C
C        If the current node has the reference 3, then the Dirichlet
C        value is 2 degC, otherwise the temperature imposed is 3 DegC
         IF (NUMREF.EQ.3) THEN            <-- For example, nodes can be distinguish by the reference number
            VDIRS(N) = 2.0D0
         ELSE
            VDIRS(N) = 3.0D0              <-- Value of the Dirichlet (W/m2)
         ENDIF
C
  300 CONTINUE
C
```

Figure 7.4: Programming of boundary conditions of the kind Dirichlet

```
C     ======================================================
C     4- INITIALIZATION OF THE BOUNDARY CONDITIONS OF THE KIND
C        HEAT EXCHANGE
C     ======================================================
C
C     Current time on the solid
      T = TEMPSS                          <-- Physical time (s)
C
C     Example ...
C
      DO 400 N=1,NBECHS                   <-- All the nodes affected by a exchange boundary condition
C
C        Global number of the node
         NUMNO = NECHS(N)                 <-- Global number of the current node
C
C        Reference of the node
         NUMREF = NREFS(NUMNO)            <-- Reference number of the current node
C
C        Coordinates of the node
         XX = COORDS(NUMNO,1)
         YY = COORDS(NUMNO,2)             <-- Coordinates of the current node
         IF(NDIM.EQ.3) ZZ = COORDS(NUMNO,3)  <-- In 3D only
C
C        TT1 = Temperature(noted TT1 here) at the current node
         TT1 = TMPS(NUMNO)                <-- Temperature of the current node
C
C        If the node has the reference 6, the external temperature
C        is set at 20 degC and the heat exchange coefficient is set
C        to 10. Otherwise, the external temperature is set to 40 degC
C        and the heat exchange coefficient is 10
         IF (NUMREF.EQ.6) THEN
            VECHS(N,1) = 20.0D0           <-- Temperature (C)
            VECHS(N,2) = 10.0D0           <-- Exchange coefficient (W/m2 K)
         ELSE
            VECHS(N,1) = 40.0D0
            VECHS(N,2) = 10.0D0
         ENDIF
C
  400 CONTINUE
C
```

Figure 7.5: Programming of boundary conditions of the kind Exchange

```
C     ======================================================
C     5- INITIALIZATION OF THE BOUNDARY CONDITIONS OF THE KIND
C        CONTACT RESISTANCE
C     ======================================================
C
C     Current time on the solid
      T = TEMPSS                    <-- Physical time (s)
C
C     Example ...
C
      DO 500 N=1,NBRESS             <-- All the nodes affected by a contact resistance
C
C        Global number of the node
         NUMNO = NRESCS(N,1)        <-- Global number of the current node
C
C        Reference of the node
         NUMREF = NREFS(NUMNO)      <-- Reference number of the current node
C
C        Coordinates of the node
         XX = COORDS(NUMNO,1)       <-- Coordinates of the current node
         YY = COORDS(NUMNO,2)
         IF(NDIM.EQ.3) ZZ = COORDS(NUMNO,3)   <-- In 3D only
C
C        TT1 = Temperature(noted TT1 here) at the current node
         TT1 = TMPS(NUMNO)          <-- Temperature of the current node
C
C        If the node has the reference 4,
C        the contact resistance is set at 0.5, and 0.7 otherwise
         IF (NUMREF.EQ.4) THEN
            VRESCS(N,2) = 0.5D0     <-- Contact resistance (W/m2 K)
         ELSE
            VRESCS(N,2) = 0.7D0
         ENDIF
C
  500 CONTINUE
C
```

Figure 7.6: Programming of boundary conditions of the kind contact resistance

7.4 Boundary conditions on faces : LIMFSO

This user subroutine is to be used when the option 'Set up of the solid boundary conditions by' is set to 'face'.

It is important to note that in that case all boundary conditions (except Dirichlet, where by definition the temperature is set on the nodes) are imposed on faces. At the subroutine level, users have to set for each face, the node values of the current face.

The principle behind the boundary conditions defined by face is similar to those exposed for the boundary conditions by nodes. Therefore, the methodology is only detailed for the flux condition. For the other conditions, the technique to use is identical, and readers can report to the previous paragraphs to know the variables introduced.

```
C     ========================================================
C     2- INITIALIZATION OF THE FLUX BOUNDARY CONDITION
C     ========================================================
C
C     Current time on the solid
      T = TEMPSS          <-- Physical time (s)
C
C
C     Example ...
C     For each face affected by a flux condition
      DO 200 N=1,NBFFLU   <-- All the faces affected by a flux boundary condition
C
C        Global number of the face
         NUMFA = NFFLUS(N)   <-- Face number
C
C        Reference of the face
         NUMREF = NREFAL(NUMFA)   <-- Reference number of the current face
C
C        There are NDMASS nodes by boundary face
C        i.e 6 in 3D, and 3 in 2D
C        For each node of the face
         DO 210 J=1,NDMASS
                                                  Loop on the face nodes
C
C           Coordinates of the nodes on  the face
            NNGLOB = NODEUS(NUMFA,J)
            XX = COORDS(NNGLOB,1)   <-- Coordinates of the cuurent node of the current face
            YY = COORDS(NNGLOB,2)
            IF(NDIM.EQ.3) ZZ = COORDS(NNGLOB,3)   <-- In 3D only
C
C           TT1 = Temperature(noted TT1 here) at the current
            TT1 = TMPS(NNGLOB)   <-- Temperature of the current node
C
C           If the face has the reference 3, the flux on the nodes
C           of the face is set to 1000 W/M2, and 10000 W/M2 otherwise
            IF (NUMREF.EQ.3) THEN
               VFFLUS(N,J) = 1000.0D0   <-- Flux for the node j of the face n (W/m2)
            ELSE
               VFLUSS(N,J) = 10000.0D0
            ENDIF
C
  210    CONTINUE
C
  200 CONTINUE
```

Figure 7.7: Boundary conditions of the kind flux set on faces

7.5 Volumetric flux : CFLUVS

The volumetric flux may depend of the time, space, local quantities like temperature, etc...

```
C     Time on the solid
      T = TEMPSS          <-- Physical time (s)
C
C     For each node with a volumic flux
      DO 100 N=1,NBFLVS   <-- All the nodes affected by a volumic flux
C
C        Global number of the node
         NUMNO = NFLUVS(N)    <-- Global node number
C
C        Reference of the node
         NUMREF = NREFS(NUMNO)   <-- Reference number of the current node
C
C        Coordinates of the node
         XX = COORDS(NUMNO,1)          Coordinates of the cuurent node
         YY = COORDS(NUMNO,2)
         IF (NDIM.EQ.3) ZZ = COORDS(NUMNO,3)   <-- In 3D only
C
C        Temperature(noted TT1 here) at the current node (NUMNO here)
         TT1 = TMPS(NUMNO)   <-- Temperature of the current node
C
C        If the current node has the refernce 4 then flux=1000 W/M3
         IF ( NUMREF.EQ.4 ) THEN
           VFLUVS(N,1) = 1000.D0
C
C        otherwise the flux is set to 10000 W/M3
         ELSE
           VFLUVS(N,1) = 10000.D0 <-- Volumic flux
C                                     (W/m3)
         ENDIF
C
  100 CONTINUE
C
```

Figure 7.8: Programming the volumetric flux

7.6 Setting references of faces : INREFA

This user user subroutine is only useful in the particular case of users having access to mesh generator without the possibility to define references on faces. Then, to apply boundary conditions on faces, one needs to create face references from the node information. This is the goal of the program *inrefa.F*.

It should be noted that programming this program can become a little bit tedious (especially in dimension 3 when very complex configurations may have to be treated). Users are therefore advised to do a careful analysis of their problem to figure out if effectively the boundary conditions by face are compulsory, or if node conditions are insufficient to handle their problem.

The principle of the subroutine is quite simple : pour a given mesh, all element are examined, and for each of them all faces are scanned. In dimension 2, the "faces" of an element are the three edges of the triangle.

For dimension 3, the principle is the same. The following figure presents the convention retained for the local numbering of vertex and edges.

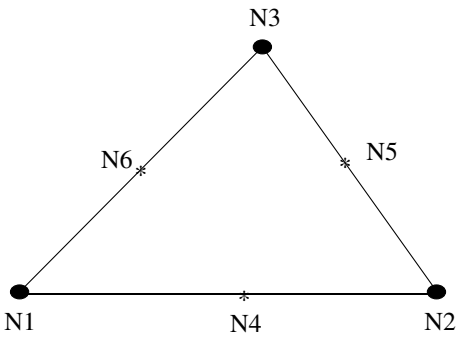

Figure 7.9: Nodes number on the face of a tetrahedron

```
C     2- CASE OF A MESH COMPOSED OF TRIANGLES
C     ================================
C
      IF (NCTHFS.EQ.3 .AND. NDIELE.EQ.2) THEN
C
        DO 200 J=1,NBFACE
          DO 210 I=1,NELEMS
C
C           Number of the 3 nodes on the edge : 1----3----2
            N1 = NODES(I,NARE(1,J))
            N2 = NODES(I,NARE(2,J))
            N3 = NODES(I,NARE(3,J))
C
C           References of the three nodes
            NR1 = NREFS(N1)
            NR2 = NREFS(N2)
            NR3 = NREFS(N3)
C
C           Coordinates of the three nodes
            X1 = COORDS(N1,1)
            Y1 = COORDS(N1,2)
              X2 = COORDS(N2,1)
              Y2 = COORDS(N2,2)
            X3 = COORDS(N3,1)
            Y3 = COORDS(N3,2)
C
C           Setting of the reference of the current edge
            NREFAC(I,J) = ???
C
  210   CONTINUE
  200  CONTINUE
C
```

For each edge of each triangle

N3 *N1* *N2*

Reference numbers of the 3 nodes of the current edge

Coordinates of the 3 nodes of the current edge

Reference to be imposed to the current edge

Figure 7.10: Set up of the references on “face” in dimension 2

7.7 Radiation user subroutine : LIMRAY

As already seen in the previous chapters, the material properties and the boundary conditions for the radiation are generally given through the graphical interface or indicated in the radiation input data file named *syrthes.ray*. Then the conditions imposed are constant in time and defined on subpart of the domain.

However, even if the situation is seldom encountered, one may need to do calculations where the radiation data are time, space, temperature, etc... dependent. In that case, the user program *limray.F* can be used. It allows to describe complex radiation behavior of the wall. Among the variables possible to set, one finds :

- the face emissivity,
- the temperature of the faces affected by an imposed temperature condition,
- the flux by band of the faces affected by an imposed flux condition,

In this user subroutine as well as the previous ones the time is accessible under the variable name TEMPSS.

7.7.1 Set up of the spectral bands

In the program *limray.F*, it is also possible to set the bounds of the spectral bands instead of using the section relative to this aspect in the graphical user interface. The section relative to this task

is detailed thereafter.

```
C     ================================================================
C     1- DEFINITION OF THE SPECTRAL BANDS
C     ================================================================
C     Definition if needed of the spectral bands
C     The number of bands (noted NBANDE) is defined in the interface
C     Example : If the number of band (NBANDE) is 2, and
C               1st band lambda_1 = 0 (m)     , lambda_2 = 5.e-6 (m)
C               2nd band lambda_1 = 5.e-6 (m) , lambda_2 = 1. (m)
CUTI          SPECTL(1,1) = 0       <-- Band 1, min
CUTI          SPECTL(1,2) = 5.e-6   <-- Band 1, max
CUTI          SPECTL(2,1) = 5.e-6   <-- Band 2, min
CUTI          SPECTL(2,2) = 1.      <-- Band 2, max
C
```

Figure 7.11: Definition of the spectral bands

One reminds users that :

- the maximum number of spectral bands admitted is 100 so far,
- the number of spectral bands has to be indicated in the radiation data file *syrthes.ray* through the graphical interface, see "NOMBRE DE BANDES SPECTRALES POUR LE RAYONNEMENT=" (Page 65)
- SPECTL(i,1) : lower bound of the spectral band i
- SPECTL(i,1) : upper bound of the spectral band i

7.7.2 Set up of the emissivities

In the user program *limray.F*, the section devoted to set the emissivity of the faces is presented below :

```
C     ================================================================
C     2- DEFINITION OF THE EMISSIVITY
C     ================================================================
C
C     Example ...
C
C     For each face of the radiation grid
CUTI  DO 200 N=1,NELRAY          <-- All the radiation faces
C
C        Reference of the current face
CUTI     NUMREF = NRFRAY(N)      <-- Reference number of the current face
C
C        Coordinates of the nodes of the current face
CUTI     XX1 = COORAY(NODRAY(N,1),1)
CUTI     YY1 = COORAY(NODRAY(N,1),2)          <-- Coordinates of the first node
CUTI     IF(NDIM.EQ.3) ZZ1 = COORAY(NODRAY(N,1),3)   of the current face
C        ...
C
C        Temperature of the current face
CUTI     TTFAC = TEMRAY(N)       <-- Temperature of the current face
C
C        Example
C        If the face has the reference 3, the emissivity is set to:
C              0.7   for the first band
C              0.95  for the second band
C        otherwise it is set to 0.5.
C        Here the emissivity could depend of the local temperature
C        (noted (TTFAC))
CUTI     IF (NUMREF.EQ.3) THEN    <-- If the reference number is 3
CUTI        EMISSI(N,1) = 0.7          Emissivity for the band 1
CUTI        EMISSI(N,2) = 0.95         Emissivity for the band 2
CUTI     ELSE                     <-- For the other faces
CUTI        EMISSI(N,1) = 0.5          Emissivity for the band 1
CUTI        EMISSI(N,2) = 0.5          Emissivity for the band 2
CUTI     ENDIF
C
CU200 CONTINUE
```

Figure 7.12: Set up of the emissivities

The loop is done on all the elements of the radiation mesh.
For a face N, one has access to:

- the coordinates of the face nodes COORAY(NODRAY(N,i),j) : designates the coordinate j of the node i of the face N. In dimension 2, $i, j \in [1, 2]$, in dimension 3 $i, j \in [1, 3]$,
- the face temperature : TTFAC, (in degrees)
- the face reference : NUMREF

One has to give :

- the emissivity for the band i : EMISSI(N,i).

7.7.3 Set up of the imposed temperature on the faces

In the user program *limray.F*, the section devoted to set the temperature of the faces affected by an imposed temperature condition is presented thereafter:

```
C     ================================================================
C     3- SETTING OF THE TEMPERATURE OF FACES WITH IMPOSED TEMPERATURE
C     ================================================================
C
C     For all faces affected by an imposed temperature condition
      DO 300 N=1,NFTIRA          <-- All the imposed temperature faces
C
C        Global number of the face
         NGFAC = NGFTIR(N)       <-- Global number of the current face
C        Reference of the current face
         NUMREF = NRFRAY(NGFAC)  <-- Reference number of the current face
C
C        Coordinates of the nodes of the current face
         XX1 = COORAY(NODRAY(NGFAC,1),1)          <-- Coordinates of the first
         YY1 = COORAY(NODRAY(NGFAC,1),2)              node of the current face
         IF(NDIM.EQ.3) ZZ1 = COORAY(NODRAY(NGFAC,1),3)
C        ...
C
C        Example
C        Warning : the temperature has to be indicated in degree C
C        If reference 5 indicates an inlet at T = 30 deg C
C        If reference 6 indicates an outlet at T = 50 deg C
         IF ( NUMREF .EQ. 5 ) THEN      Depending on the reference number :
            TEMRAY(NGFAC) = 30.         <-- imposed temperature is 30 C
         ELSEIF ( NUMREF .EQ. 6 ) THEN
            TEMRAY(NGFAC) = 50.         <-- imposed temperature is 50 C
         ENDIF
C
  300 CONTINUE
C
```

Figure 7.13: Set up of the radiation imposed temperature

The loop is done on the faces affected by the corresponding condition : (see page 66)
For a face N, one has access to:

- the global number of the face : NGFAC,
- the reference of the face : NUMREF,
- the coordinates of the face nodes COORAY(NODRAY(N,i),j) : designates the coordinate j of the node i of the face N. In dimension 2, $i, j \in [1, 2]$, in dimension 3 $i, j \in [1, 3]$,

One has to give :

- the temperature to set on the face : TEMRAY(NGFAC)
  (the temperature has to be given in degree Celsius)

7.7.4 Set up of the radiation imposed flux

In the user program *limray.F*, the section devoted to set the imposed flux on faces is presented thereafter :
The loop is done on the faces affected by the corresponding condition : (see page 66)

For a face N, one has access to:

- the global number of the face : NGFAC,
- the reference of the face : NUMREF,

- the coordinates of the face nodes COORAY(NODRAY(N,i),j) : designates the coordinate j of the node i of the face N. In dimension 2, $i, j \in [1, 2]$, in dimension 3 $i, j \in [1, 3]$,

One has to give :

- the flux to be imposed on the face N for the band i : VFIRAY(N,i,1)
  (The flux must be given in W/m^2).

```
C     ================================================================
C     4- SETTING OF THE FLUX OF FACES WITH IMPOSED FLUX BY BAND
C     ================================================================
C
C     For all faces affected by an imposed flux condition
      DO 400 N=1,NFFIRA          <-- All the imposed flux faces
C
C        Global number of the face
         NGFAC = NGFFIR(N)       <-- Global number of the current face
C        Reference of the current face
         NUMREF = NRFRAY(NGFAC)  <-- Reference number of the current face
C
C        Coordinates of the nodes of the current face
         XX1 = COORAY(NODRAY(NGFAC,1),1)
         YY1 = COORAY(NODRAY(NGFAC,1),2)      <-- Coordinates of the first
         IF(NDIM.EQ.3) ZZ1 = COORAY(NODRAY(NGFAC,1),3)   node of the current face
C        ...
C
C        Example
C        One has only one band --> NUMBAN = 1
C        If the reference of the face is 7, one has an adiabatic face
C        If the reference of the face is 3, one has a flux of 500 W/m2
         NUMBAN = 1
C
         IF ( NUMREF .EQ. 7 ) THEN      <-- Depending on the reference number :
            VFIRAY(N,NUMBAN,1) = 0.     <-- imposed flux is 0 W/m2K
         ELSEIF ( NUMREF .EQ. 3 ) THEN
            VFIRAY(N,NUMBAN,1) = 500.   <-- imposed flux is 500 W/m2K
         ENDIF
C
  400 CONTINUE
C
```

Figure 7.14: Set up of the radiation imposed flux

Part II

SYRTHES - Performing

Methodology to perform a calculation 8

SYRTHES has been conceived so that the methodology to perform a SYRTHES calculation be the same if one has to deal with a purely conductive study or a study where the solid wall interacts with a fluid.

Whatever, the kind of applications aimed at, it is always advised to begin with simple studies where only conduction or radiation take place. Even if the final goal is to perform coupled studies, learning to use SYRTHES alone is always worth the trouble, since the methods and the parameters and options stay the same at the SYRTHES level. Indeed, for SYRTHES, one has to recall that the fluid constitutes only a particular boundary condition.

When used as the solid module of a fluid/solid calculation, one has to remember that SYRTHES is completely decoupled from the CFD code, and that the two codes can be analyzed almost separately. It can even be interesting, in the case of particularly complex studies, to start with only the fluid calculation to identify the potential difficulties. Then, in a second step, the coupled calculation can be done with any lost of time or energy[1].

Likewise, if a user runs into numerical trouble, one has always to remember that it is always possible to decouple SYRTHES and the CFD code, only by setting the SYRTHES activation keyword to **No**. Such a procedure may, in some cases help to identify the origin of a problem. Is it a purely fluid one, a solid one, or is it related to the data transfer between the solid code and the CFD code, etc...

In this chapter, one will describe the generally followed method to perform a calculation. In particular, the different steps will be detailed as well as their chronology.

In the case of coupled fluid/solid calculation, the notions to run the fluid code are not described in the present document, users are advised to refer to the CFD users manuals.

8.1 Methodology to perform a SYRTHES calculation

After listing the main steps in a calculation, some of them will be revisited to insist on key points. The different phases of a calculation are the following :

- analysis of the physical problem, choice of the calculation domain, of the physical models,

[1] However, one has to note that the behavior of a flow can be affected if the fluid calculation takes or not the thermal flux on the wall boundary

- generation of the non structured grid, setting of the references to identify the different boundary conditions, the material properties, etc...

  and if necessary, the generation of the radiation mesh if the user intend to take the radiation exchange into account.

- setting of the conduction and radiation parameter files through the EDAMOX interface.
- setting (if necessary) the user subroutines for the treatment of :

  ▷ initial conditions,

  ▷ material properties,

  ▷ boundary conditions,

  ▷ volumetric flux,

  ▷ radiation.

- setting of the main program to dimension the real and integer main vectors.
- writing the organization file *syrthes.env* and the batch file (according the computer used)
- running of the program
- and hopefully analyzing the results ...

8.1.1 Performing a calculation SYRTHES-CFD code

To get an optimal use of this numerical tool, users are advised to prepare and organize their problem as much as possible. Often this preliminary work turns out to be very rewarding afterwards. This rule is already true for simple CFD codes, but it becomes crucial when the problem requires a fluid/solid calculation since it brings along more files and physical mechanisms. The great flexibility provided by the SYRTHES module, the independence of its files relative to the fluid files should allow a quick understanding for users already knowing how to manage a fluid calculation.

For a complex problem, where the fluid/solid thermal coupling functionality is required, one may suggest two phases :

- **1st step : fluid calculation only with thermal aspects**

  Perform a purely fluid calculation without any coupling with a solid. It is therefore a standard fluid calculation. However, set a thermal wall condition on the nodes supposed to be affected by the solid code. The best is of course to impose wall temperature (close to the one expected in the coupled case, but of course constant in time), if it is possible in the fluid code.

  Once coherent and physical results obtained (in this preliminary and intermediate test allowing to progress in the study analysis), the real coupled problem can be handled.

- **2nd step : thermally coupled calculation SYRTHES - CFD code**

  The previous phase, has checked the good behavior of the fluid code. The time devoted to the first phase can now entirely be reused. Indeed, all parameters and files generated for the first phase are ready. The only change in the fluid parameter file should be to activate the SYRTHES module thanks to a unique keyword (the only difference from the user point of view with a standard fluid calculation).

8.2 Coupling *Code_Saturne*- SYRTHES: "impact" on the fluid side.

As already mentioned in this document, the fluid and solid problems are treated independently. The only impact on the fluid calculation is located on the boundary condition to set on the fluid boundary nodes to be coupled with SYRTHES.
Here the procedure may differ from one CFD code to another.

8.3 Generating meshes for SYRTHES

The solid domain is spatially discretized thanks to an unstructured mesh. So far, the code handles the data base coming from the mesh generator SIMAIL [14] or IDEAS-MS [15].

During the mesh generation phase, users have to keep in mind their physical problems. They must remember that :

- some zone more sensitive than other, where important physical phenomena take place, have to be meshed more carefully than others.
- generating regular elements improves the calculation convergence. One should in particular prevent as much as possible elements having inner angles greater than 90°C, and a reasonable distortion should be kept.
- already during the meshing step, physical characteristics of the material have to be taken into account. Indeed, the references on nodes or element have to be chosen so that it be easy afterwards to identify all the different materials. Likewise, the boundary nodes or faces have to be carefully referenced to distinguish :
  - ▷ the entities potentially coupled with the fluid,
  - ▷ the entities affected by flux conditions,
  - ▷ the nodes affected by Dirichlet conditions,
  - ▷ the entities affected by exchange conditions,
  - ▷ the entities affected by contact resistance,
  - ▷ the entities affected by volumetric flux,
  - ▷ Belonging to different materials
  - ▷ the entities coupled to radiation

It is important to notice that references have to be chosen between 1 and 99. A node whose reference would be 0 (zero) would be considered without reference.

8.4 Running a calculation

8.4.1 Creating a case of calculation

A utility is available to automatically create " case calculation", ie a directory containing a copy of all files that may be Useful for the calculation. This allows for example to have a data file pre-filled where only the values of some parameters will change.

```
syrthes_create_case
```

8.4.2 Case of a SYRTHES calculation

Once the data file updated, and the eventual user subroutines completed, users have to enter

```
link_syrthes
```

which is in charge of compiling the user subroutines and do the link procedure to generate the executable named *syrthes*.

Then, the code can simply be run by the following command :

```
syrthes > listing &
```

Here, the syrthes code is run in batch and the standard output is redirected in a file named "*listing*".

8.4.3 Case of a calculation *Code_Saturne*-SYRTHES

The data files are prepared independently for the two codes. Then, this is indicated in the file to launch *Code_Saturne* to be taken into account the coupling with syr. The user will refer to procedure Launch sat [2] for details of the operation.

8.5 Analysis of computation results

Analyzing a calculation should always beginning by a thorough look at the listing file.

Especially for the first runs, one should check carefully the information given by the code during the initialization phase. For example, the number of nodes, elements, affected by a special conditions, etc... should be looked at.

These initial verifications have several purposes:

- a possibility for users to control their data : check if all conditions have been accounted for, if references and conditions are coherent, if the boundary conditions values are exact, etc...
- ensure that there is a good adequation between what was intended and what has been interpreted by the code,
- check that the initializations steps do behave correctly.

During the run
Within each time step, it is possible to get information on the level of convergence of the solver (printing level above 2). When a stationnary state is looked after, the number of iterations required by the solver should gradually get smaller as time goes by if the calculation behaves correctly.

The solver convergence
If convergence difficulties do appear (high number of iterations for a poor level of convergence), it is generally possible to improve the situation by reducing the time step (this leads numerically to a better system conditioning).

In the case of transient calculations, it is preferable to get a good level of convergence at each time step. If the precision reached stays intermediate, one suggests to increase the maximum number

of iterations allowed or reduce the time step used. This is even more important to do so if surfacic flux and volumetric flux are present in the calculation (since they are treated explicitly).

How to appreciate convergence
In all numerical approaches, appreciating convergence stays tricky even if it often represents a main interest for users. Indeed, no absolute criteria ensure that a specific problem has converged. A certain number of indicators may however bring some help, notably :

- the behavior of the iterative solver
- the use of thermal probes in some well chosen locations. Analyzing the temperature evolution of these nodes show if an asymptote has been reached or not.
- the analysis of thermal contours in a post-processor, recorded at several time steps,
- and last but not least how experienced the user is...

8.6 Troubleshooting methodology

This paragraph proposes a strategy to overcome problems to which users may be confronted. When errors occur, it is important to follow a logical trouble shooting methodology to identify and hopefully solve the problem. What was the first symptom, or under what consistent conditions does the problem occur? Can you narrow down the problem to a smaller set of conditions to reproduce the problem? Can you work around the problem or is it a blocking problem? Do you think you can manage yourself or do you need Hotline support? Should the problem be reported so that it can be fixed in a future release?

In most cases, the first step is to analyze the nature of the misbehavior. According the conclusion, a methodology is proposed to focus on the real problem and actions to be taken to overcome it.

8.6.1 The great family of troubleshooting

In the following sketch, diverse kinds of troubleshooting are presented and the actions to take if one recognizes his own type of problem.

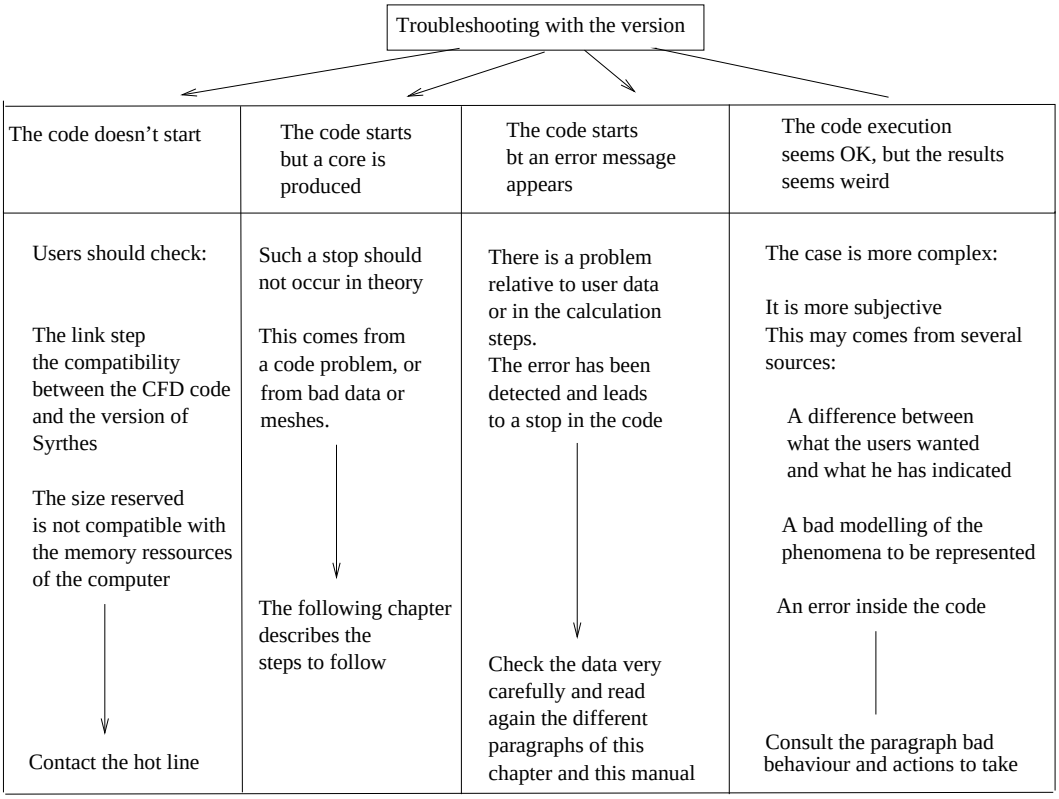

Figure 8.1: Classification of the troubleshooting

8.6.2 Unwanted stop during the run

Especially for complex numerical codes, an unwanted stop of the code may arise. This may have multiple origins. It is frequent then to be confronted with a division by zero, numbers too large or too small for the computer to handle, wrong arguments for a mathematical function, etc...

The first action to take is then to check very carefully the data used (keyword, boundary conditions, user subroutines). Consider all the variables that might be having an effect. Run the code on a different computer (even if softwares are usually fairly deterministic, meaning that the same input yield the same result), but you may not be aware of all input variables (like memory, disk space, other programs running at the same time, etc...)

If the problem should subsist, collect all available clues, including error messages, error output files, message written, etc... and then contact the hot-line.

8.6.3 What to do if a SYRTHES error message appears

The numerical codes in question are very complex, and requires numerous data. To guaranty as much as possible reliable results, many tests are done to check the coherence of the data introduced and prevent problems afterwards. Likewise, many numerical tests are done during the calculation to prevent a computer core to be generated.

Then the code is stopped properly, and in most cases, the two conceptors of SYRTHES have tried to give an explanation as clear as possible of what has motivated the stop of the code, and give some insights of what actions should be taken to remedy this situation. The following sketch summarizes the actions to take when faced with such a situation.

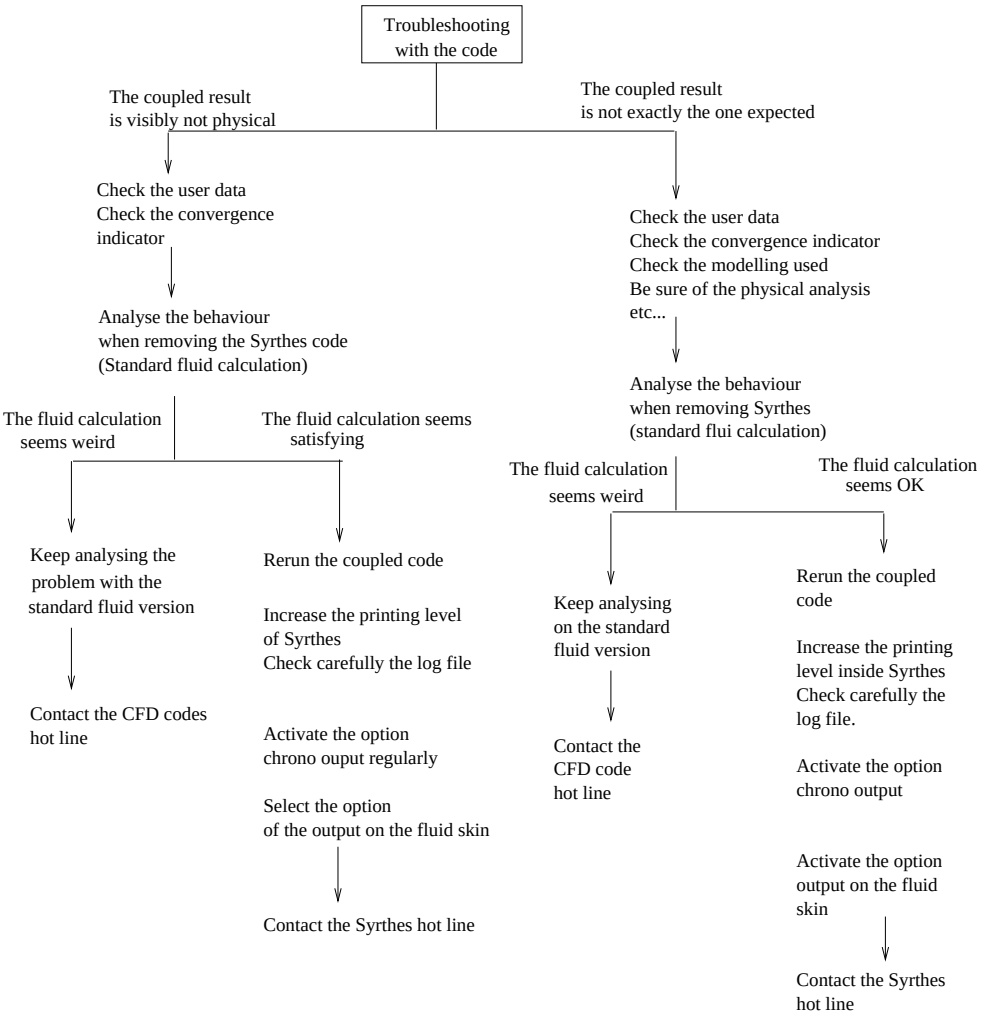

Figure 8.2: Methodology to follow when faced with a error message

8.6.4 What to do in case of a "bad" simulation?

Users are generally confronted to complex systems. Once the numerical results obtained, even if the code has run without any apparent problem, a first critical analysis may show strong differences with respect to what was expected. And one may question the validity of the numerical approach.

It is clear that such a kind of problem is much more difficult to handle than "simple" crash cases. Indeed, the "quality" of a numerical result is an extremely difficult notion to define, moreover, in some situations, it appears to be partly subjective, in the most complex problems. Validation procedures performed on the CFD codes as well as SYRTHES allow to reduce risks of this kind and allow to get a better grasp of the range of validity of these numerical tools.

Even if it is rare, it may happen that numerical results be very far from the expected answers. How users should react when faced with such a situation.

If it is question of a thermally coupled problem, one must begin to identify if the "weird" behavior originates from the fluid side or the solid side. This can be achieved easily by decoupling SYRTHES. Doing this is almost instantaneous, indeed commenting or deactivating the proper keyword or in the fluid parameter file does the job. It is easy to understand that this operation limits already considerably the field to investigate.

The following sketch summarizes a possible methodology to follow.

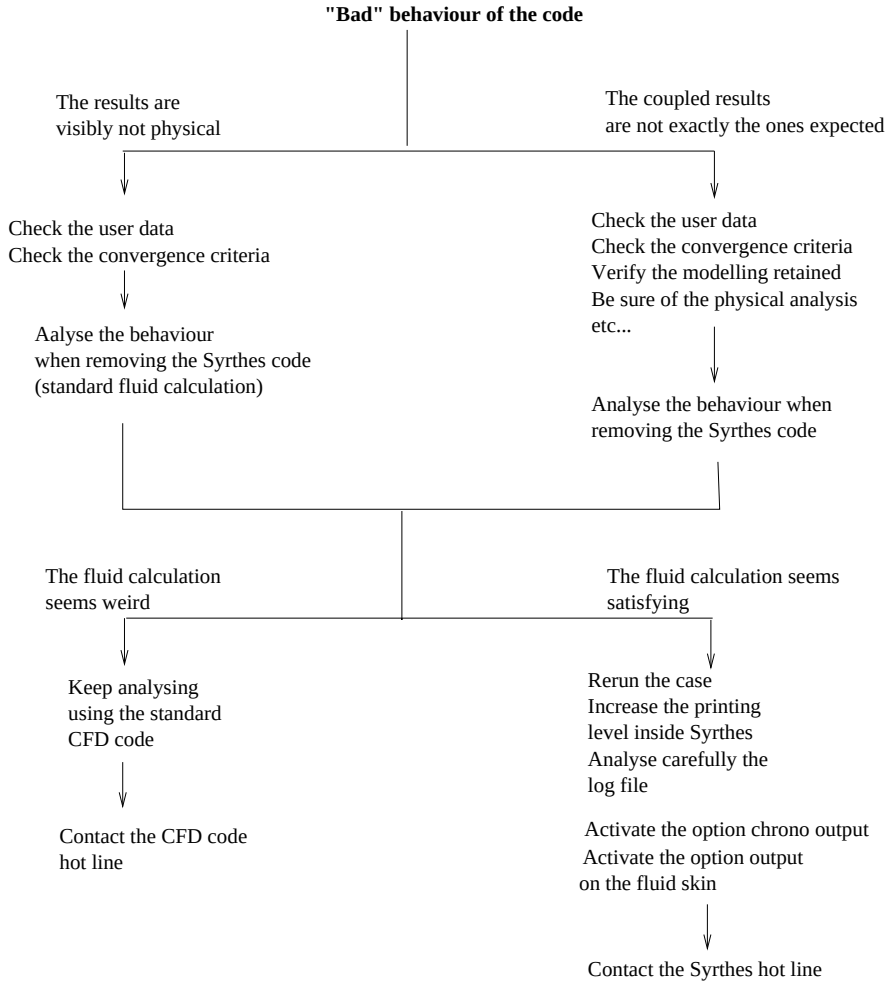

Figure 8.3: Methodology to follow faced with a bad behaviour

Examples 9

The main goal of this chapter is to describe several elementary examples and underlines how they should be handled thanks to SYRTHES. It should be noted that the fluid part is not the subject treated here when thermal fluid/solid coupling takes place. : users are encouraged to refer to the CFD codes own users manual.

In this document, 8 problems illustrating part of the functionalities of SYRTHES are described. The first case named *3rond2d* is quite didactic to help new users getting familiarized with the code SYRTHES. Consequently, the case is treated thoroughly and a step by step presentation has been adopted. Users are guided through the calculation. Concerning the problems following *3rond2d*, one supposes that users have already some basis. Therefore, voluntarily only the specifities of each case are presented. Users will always have the possibility to go back to the first example given if necessary.

The illustrating problems presented are the following :

- **3rond2d** : problem of a thermal transient calculation involving different conductivity behavior (isotropy, orthotropy, anisotropy)
- **Periodicity** : problem involving periodicity in two direction. One simulates a very large plate with holes.
- **Contact resistance** : problem involving contact resistance (to simulate a crack).
- **3rond2d_ray** : problem where radiation from wall to wall has to be taken into account.

It should be underlined that the only purpose of the examples gathered in this document is to illustrate some functionalities of SYRTHES and how to use them. Even if some cases appear as similar to industrial cases to which users could be confronted with, no conclusions should be drawn from the present examples.

The different problems are commented more or less according the specifities of each case. This may lead to say again many things already discussed previously. On the contrary, one may hope it will help users getting a better understanding on the way to use SYRTHES efficiently.

9.1 Getting started with a solid thermal problem

Solids with anisotropic behavior
3rond_2d

9.1.1 Case description - Problem analysis

The solid domain is constituted of three separate disks, each affected by a different (anisotropic) conductivity :

- For the disk 1, one sets a conductivity equal along axis x and along axis y (25 W/mK);
- For the disk 2, the conductivity along the direction x is set to 25 W/mK, while it is only 5 W/mK along the axis y.
- For the disk 3, the conductivities are 25 W/mK and 5 W/mK along the axis of a local system of coordinates being at a 45° with respect to the system of coordinates of reference.

The density and the specific heat are considered identical for the three disks and set at : $\rho = 7700\ kg/m^3$ and $C_p = 460\ J/kgK$.

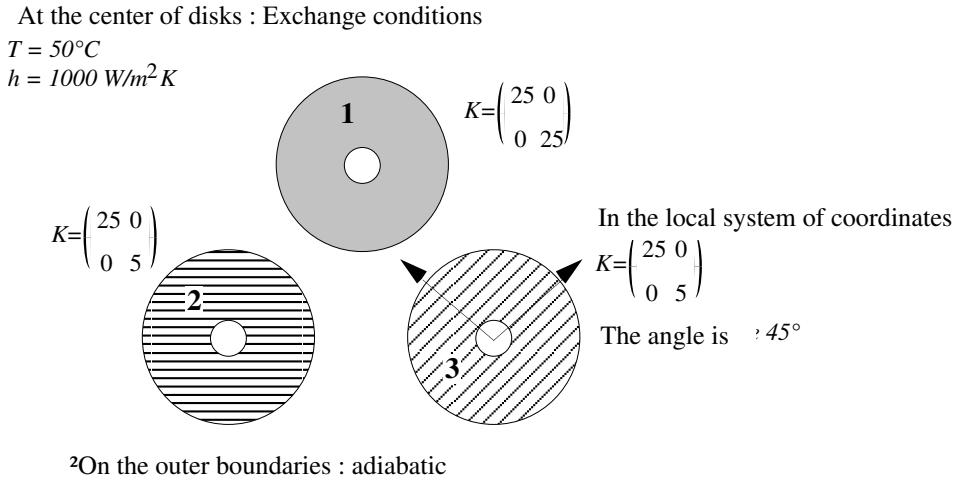

Figure 9.1: Sketch of the problem

9.1.2 Generation of the mesh

If the geometry stays very simple and do not create any trouble, attention must however be paid to the references allowing to identify materials and boundary conditions.

In the present case, one has chosen to use nodes references. The nodes of each disk have to carry different references. Moreover, inside each disk, one has to identify the nodes affected by an exchange condition.

It should be noted that the notion of adiabaticity (zero flux) being implicit in the code, there is no need to identify the corresponding nodes by a specific reference.

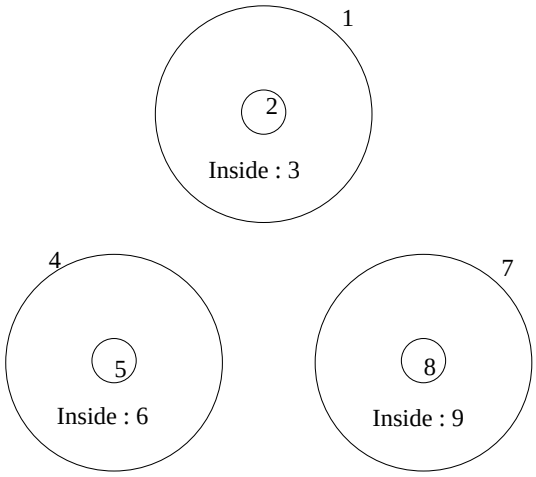

Figure 9.2: Setting the references

The conduction mesh contains 11688 nodes (P_2 discretization) and 5688 elements.

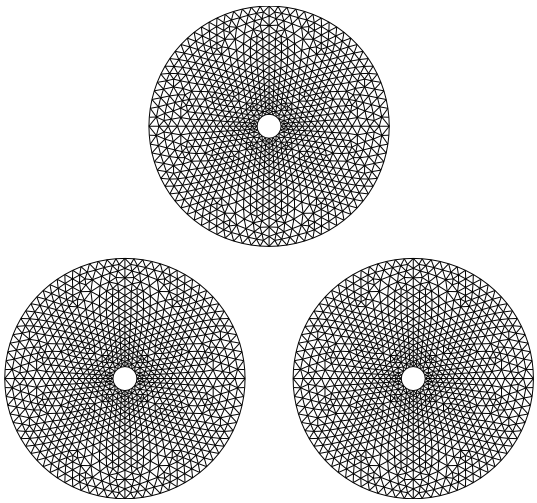

Figure 9.3: Conduction mesh

9.1.3 The parameter file

9.1.3.a Generalities

The problem is in dimensin 2, and it is not an axisymmetric case. It has been decided that boundary conditions whould be set on nodes.

```
'DIMENSION DU PROBLEME=' 2
'AXE D AXISYMETRIE (AUCUN,OX,OY)=' 'AUCUN'

'CONDITIONS LIMITES PAR NOEUD OU PAR FACE='  'FACE'
'DEFINITION DES PROPRIETES PHYSIQUES PAR=' 'ELEMENT'
'DEFINITION DES FLUX VOLUMIQUES PAR=' 'ELEMENT'
```

9.1.3.b Computation management

We have to set the time step since we are dealing with a thermal transient (the converged solution is of no interest, here all disks will reach a uniform temperature). The time steps has to stay reasonnable if a fair precision is required during the transient, indeed the time error is more or less proportionnal to the time step retained. In the present case, and considering the mesh, a maximum time step from 10s to 100s seems to be reasonnable. One sets the time step to 100 (in second of course).

```
'PAS DE TEMPS SOLIDE='   10.
'NOMBRE DE PAS DE TEMPS SOLIDES=' 100
'SUITE DE CALCUL=' 'NON'
/
```

9.1.3.c Output management

```
/ Sorties
/--------
'NIVEAU DES IMPRESSIONS POUR LE SOLIDE=' 2
'PAS DES SORTIES CHRONO SOLIDE='    25
'ECRITURE MAILLAGE SOLIDE='     'OUI'
'HISTORIQUES CONDUCTION='            'OUI'
'CHAMP DE TEMPERATURES MAXIMALES='   'NON'

/ Historiques
/ -----------
'HIST' 'FREQ' 100.
'HIST' 'NOEUDS' 1 34 143 4
/
```

For the standard log (listing file) the printing level has been chosen as normal (which is a reasonnable choice for a first run). This level can be increased if problem should appear. Concerning the output file, users will get of course the temperature field corresponding to the last time step, but also intermediate results every 25 time steps. These temperature fields will be recorded chronologically in a "chrono" file. Like the classical result file, this chrono file can be processed by usual post-processors. One has also decided to record at higher frequency the temperature at some chosen nodes (sort of numerical thermal probes). A thermal probe result file will be generated, and will contain the temperature of theses nodes every 100 secondes, i.e every time step (since it is the frequency chosen). Comparatively, the temperature field will be recorded every 2500s.

9.1.3.d Numerical options

One should note that a much larger time step would maybe require to increase the maximum number of iterations allowed to keep an acceptable level of convergence.

```
/ Choix numeriques
/-----------------
'NOMBRE ITERATIONS SOLVEUR SOLIDE='   100
'PRECISION POUR LE SOLVEUR SOLIDE=' 1.E-6
```

9.1.3.e Material properties

At the mesh level, the entities referenced are nodes. The material properties will therefore use nodes references to be set. Here, since at least one material behaves anisotropically, one has to retain that option.

```
'ISOTROPIE DU MATERIAU='   3
/
'CPHY'     'RHO'           7700.       -1
'CPHY'     'CP'            460.        -1
/
CPHY'     'K ANISOTROPE'     25. 25. 25.  0. 0.  0.      1
CPHY'     'K ANISOTROPE'     25.  5. 25.  0. 0.  0.      2
CPHY'     'K ANISOTROPE'     25.  5. 25.  0. 0. -45.     3
```

The density is constant throughout the complete domain. Following the convention chosen, the list of references is reduced to -1. This is also the case for the specific heat. On the contrary the conductivity differs according the disk considered :

- upper disk : in fact the behaviour is isotropic. This is achieved by setting an equal conductivity along both axis.
- the left disk : the behaviour is orthotropic, the conductivity along axis x is 5 times higher than the conductivity along axis y.
- right disk : this time the conductivity is fully anisotropic. The conductivity is defined by two conductivity values along local axis of coordinates, then the angle (expressed in degrees) between the local system of coordinates and the global system of coordinates of reference.

9.1.3.f Initial conditions

Initially the temperature is set to $20°C$ on all the domain, which explains the list of references reduced to -1.

```
'CINI'          20.                -1
```

9.1.3.g Boundary conditions

At the center of the disks, a temperature of $50°C$ and a heat exchange coefficient of 1000 W/m^2K are imposed. On the outer side, the condition by default of SYRTHES is used (i.e. adiabatic), therefore no condition needs to be specified. Only the section "exchange" needs modification.

```
'REFERENCES NOEUDS OU FACES SOLIDES AVEC COEFFICIENT D ECHANGE' 2 5 8

'CLIM'    'COEF ECH'         50.  1000.       -1
```

On should note that the reference list is reduced to -1 since the numerical values (h=1000 et T=50) stay the same on all nodes affected by an exchange condition. Putting the list 2 5 8 whould give the same effect.

9.1.3.h Files for the calculation

The indications given in that section will allow to generate the organization file *syrthes.env*. This file will contain the names of the files necessary to run SYRTHES. On reminds that this file must necessarily be placed in the same directory as the executable (often name *syrthes*).

The aim of the upper part of the window is to indicate the different paths giving access to the files. These paths may be given in a relative way, from the current position in the file system. In the present case, all files are defined relatively to the current directory "(./)".

```
*
************************************************************************
*           EMPLACEMENT ET NOM DES FICHIERS DU MODULE SYRTHES           *
************************************************************************
*
*
EMPLACEMENT DES FICHIERS DU MODULE SYRTHES
  AMONT : ./
  SUITE : ./
  AVAL :  ./
*
*
NOM DES FICHIERS AMONT POUR SYRTHES
  DONNEES DU CALCUL : syrthes.data
  GEOMETRIE SOLIDE : 3rond2d.des
*
*
NOM DES FICHIERS SUITE POUR SYRTHES
  SUITE SOLIDE RESU : resus1
*
*
NOM DES FICHIERS AVAL POUR SYRTHES
  RESU SYRTHES 1 : geoms
  RESU SYRTHES 2 : resus1
  CHRONO SYRTHES 2 : resusc1
  HISTORIQUE SOLIDE RESULTAT : histos1
```

9.1.4 Running the code

In the present version, SYRTHES uses two main vectors (a real and an integer one) to manage the memory. These two tables are located in the main program *syrthes.F*. In order to optimize the required memory, users may have to adjust the size of these two tables. To give an estimation of the necessary memory, SYRTHES needs for a typical conduction study around :

- approximatively 14 integers per node P_2 of the solid mesh,
- approximatively 40 reals per node P_2 of the solid mesh.

During the initialization phase, and also during the first time step, SYRTHES indicates its memory requirements. With the information given, users may adjust the memory size specified.

A shell *link_syrthes* allows to compile the subroutine *syrthes.F* (and possibly other user subroutines) and does the linking operation to generate an executable.
To start up SYRTHES users only have to type :

```
syrthes > listing &
```

at the operating system prompt in a window. This will redirect the standard output of the code in the file *listing*.

9.1.5 Analyzing the results

Users may refer to the chapter "methodology" for the first advices regarding the results analysis. One tries in that paragraph to underline notions relative to the exploitation of calculations.

9.1.5.a The listing file

Among the different possible output, the log file *listing* which is displayed on screen or more reasonnably in a file, may be very useful to consult. It gathers numerous information relative to the way the code is behaving. A good attitude, when the run is finished, is to have a carefull look at the listing file. This allows to control that no errors has been introduced in the data file, or that the case treated is effectively the one wanted by the user. The listing allows also to control how well the solver converges, the CPU time necessary, etc...

A few elements are presented thereafter. For clarity reasons, the informations are displayed in three blocks. In this document, they appear in several sketches, but the chronological order in which they appear has been respected.

The first block is relative to the data entered by users to define the case. Generally, the time spent to control these data is worth the possible trouble it may prevent.

Printing out these informations also has the advantage to keep in memory the exact configuration studied (and sometimes a long time after).

The first part of the listing corresponds to all the initialization steps, and the preliminary calculations.

```
    ************************************************************************
    *                                                                      *
    *   SSSSS  YY    YY  RRRRRR  TTTTTTTT HH   HH  EEEEEE   SSSSS  *
    *  SS       YY  YY   RR   RR    TT    HH   HH  EE      SS      *
    *  SS        YYYY    RR   RR    TT    HH   HH  EE      SS      *
    *   SSSS      YY     RRRRRR     TT    HHHHHHH  EEEE     SSSS   *
    *      SS     YY     RR  RR     TT    HH   HH  EE          SS  *
    *      SS     YY     RR   RR    TT    HH   HH  EE          SS  *
    *  SSSSS      YY     RR   RR    TT    HH   HH  EEEEEE  SSSSS   *
    *                                                                      *
    ************************************************************************

                                VERSION 3.1

 ** CPU TIME BEFORE INITIALIZATION STEP :   0.00000
*
**************************************************************************
*           DIRECTORIES AND NAMES OF THE SYRTHES FILES                   *
**************************************************************************
*
*
DIRECTORIES FOR THE SYRTHES FILES
  INPUT : ./
  RESTART : ./
  OUTPUT :  ./
*
*
NAMES OF THE INPUT SYRTHES FILES
  CONDUCTION DATA : syrthes.data
  SOLID MESH : Simail/3rond.des
  RADIATION DATA :  syrthes.ray
  RADIATION MESH : ray.des
*
*
NAMES OF THE RESTART SYRTHES FILES
  RESTART FILE : resus1
  STORING OF THE FLUID/SOLID CORRESPONDENTS :  corresp
  RADIATION VIEW FACTORS : fdf
  STORING OF THE RADIATION CORRESPONDENTS : corresp.ray
*
*
NAMES OF THE OUTPUT SYRTHES FILES
  RESU SYRTHES 1 : geoms
  RESU SYRTHES 2 : resus1
  CHRONO SYRTHES 2 : resusc1
  CONDUCTION PROBES : histos1
  FLUID BOUNDARY MESH :  ff1
  FLUID BOUNDARY RESULT : ff2
  FLUID BOUNDARY CHRONO : ff2c
  RADIATION MESH : ray.geo1
  RADIATION RESULT : ray.res1
  RADIATION CHRONO : ray.chro1
*
```

Figure 9.4: List of the files used for the calculation

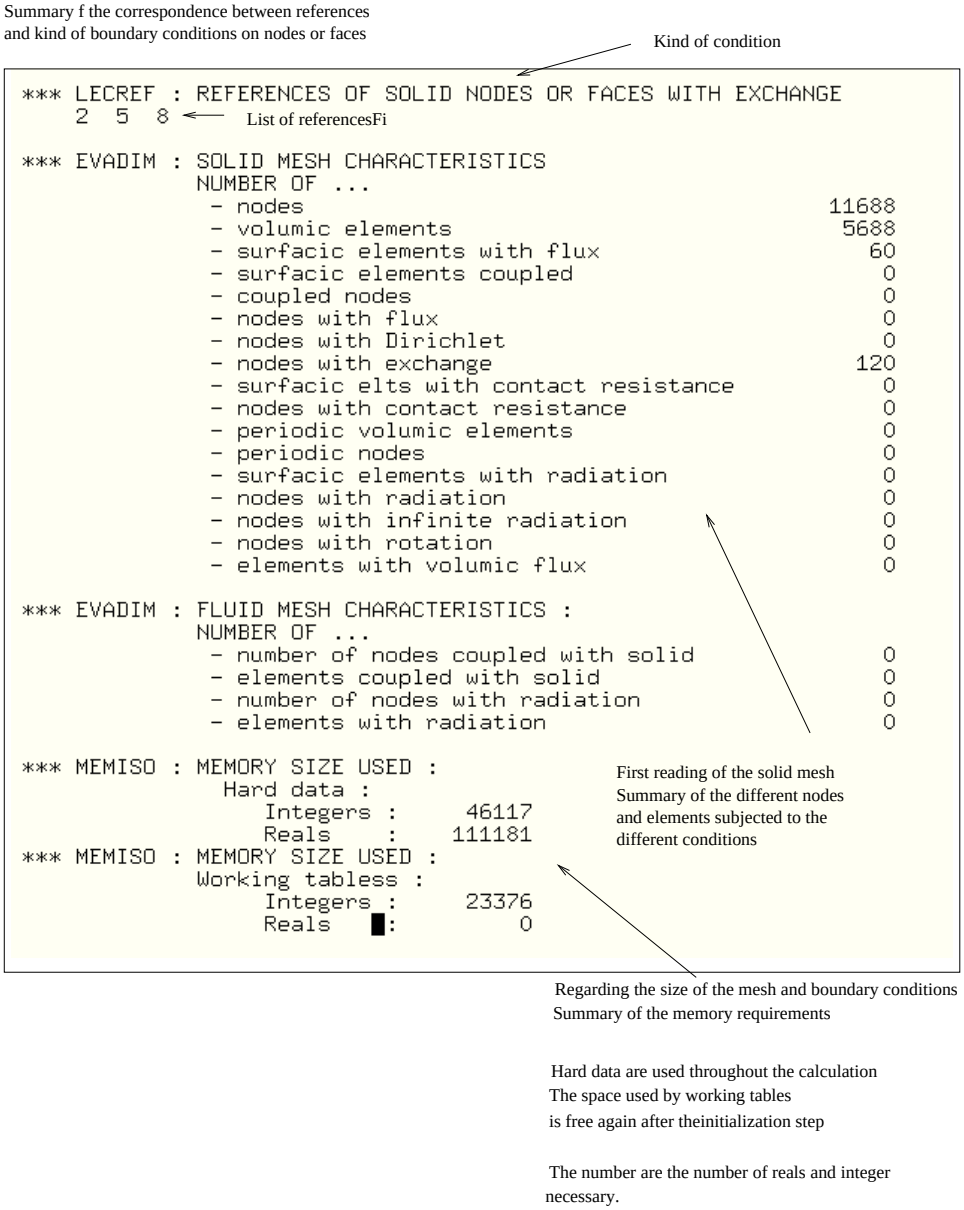

Figure 9.5: Geometrical information

```
*** LECSI2 : SOLID MESH :
       - Dimension                          :      2
       - Number of P1 nodes                 :   3000
       - Number of nodes                    :  11688
       - Number elements                    :   5688
       - Maximum of the reference numbers   :      9


*** LECSI2 : Verification of the solid mesh


             Coordinates of the 10 first nodes :

             N= 1   COORD :  0.10000E+00 0.00000E+00
             N= 2   COORD :  0.14147E+00 0.00000E+00
             N= 3   COORD :  0.18505E+00 0.00000E+00
             N= 4   COORD :  0.23083E+00 0.00000E+00
             N= 5   COORD :  0.27895E+00 0.00000E+00
             N= 6   COORD :  0.32950E+00 0.00000E+00
             N= 7   COORD :  0.38262E+00 0.00000E+00
             N= 8   COORD :  0.43844E+00 0.00000E+00
             N= 9   COORD :  0.49709E+00 0.00000E+00
             N=10   COORD :  0.55871E+00 0.00000E+00

             The 10 first elements :

             N= 1   NODES :   1000   804   803  3001  3002  3003
             N= 2   NODES :   1000   803   790  3003  3004  3005
             N= 3   NODES :    999   997   998  3006  3007  3008
             N= 4   NODES :    808   278   279  3009  3010  3011
             N= 5   NODES :    999   998   909  3008  3012  3013
             N= 6   NODES :    998   997   995  3007  3014  3015
             N= 7   NODES :    998   995   996  3015  3016  3017
             N= 8   NODES :    997   994   995  3018  3019  3014
             N= 9   NODES :    999   894   903  3020  3021  3022
             N=10   NODES :    996   995   992  3016  3023  3024

             References of the 10 first nodes :

             N= 1   REFERENCE :   2
             N= 2   REFERENCE :   3
             N= 3   REFERENCE :   3
             N= 4   REFERENCE :   3
             N= 5   REFERENCE :   3
             N= 6   REFERENCE :   3
             N= 7   REFERENCE :   3
             N= 8   REFERENCE :   3
             N= 9   REFERENCE :   3
             N=10   REFERENCE :   3
```

In order to control the mesh used: One indicates here:

The coordinates of the ten first nodes

The connectivity of the ten first elements

The references of the ten first nodes

Figure 9.6: Geometrical information

Generation of the surfacic meshon which
boundary conditions are applied
Here by convention, the flux elements coresponds to:
Imposed flux
Exchange conditions
Contact resistance

```
*** XMAILL : SURFACIC SOLID MESH :
       - Number of surfacic coupled elements     :      0
       - Number of surfacic flux elements        :     60
       - Number of surfacic resistance elements  :      0
       - Number of surfacic radiation elements   :      0

*** LIMNUM : Number of solid nodes :
       - coupled with fluid          :      0
       - with flux condition         :      0
       - with Diriclet condition     :      0
       - with exchange condition     :    120
       - with contact resistance     :      0
       - with radiation              :      0
       - with infinite radiation     :      0
       - periodic                    :      0
       - moving                      :      0
```

In our example, the boundary conditions
are applied on nodes
Here is a summary of the number of nodes
affected by different kind of conditions

Remark:
For the option (boundary conditions on faces)
a similar summary is proposed for faces

Figure 9.7: Boundary condition

Reading of the boundary conditions and material properties in the user data file (here syrthes.data)
Each time a line has been entered through Edamox, here is how the code Syrthes has interpreted the command
This allows :
To control the boundary conditions
To keep in mind the boundary conditions which have effectively been used for the present calculation

```
*** LECLIM : Exchange boundary condition
             Temperature :  0.50000E+02    Coefficient :  0.10000E+04
             References :   2  5  8

*** LECLIM : Mass
             Value :  0.77000E+04    References :  -1

*** LECLIM : Specific heat
             Value :  0.46000E+03    References :  -1

*** LECLIM : Anisotropic conductivite
             K11,K22,K33 :  0.25000E+02 0.25000E+02 0.25000E+02
             TETA X, TETA Y, TETA Z : 0.00000E+00 0.00000E+00 0.00000E+00
             References :   1  2  3


*** LECLIM : Anisotropic conductivite
             K11,K22,K33 :  0.25000E+02 0.50000E+01 0.25000E+02
             TETA X, TETA Y, TETA Z : 0.00000E+00 0.00000E+00 0.00000E+00
             References :   4  5  6


*** LECLIM : Anisotropic conductivite
             K11,K22,K33 :  0.25000E+02 0.50000E+01 0.25000E+02
             TETA X, TETA Y, TETA Z : 0.00000E+00 0.00000E+00-0.45000E+02
             References :   7  8  9
```

Figure 9.8: Calculation conditions issued from the user data file

```
*** MEMSOL : MEMORY SIZE USED :
                Working tables :
                   Integers :            0
                   Reals    :       167136
```

At the end, the last step of the initializations

There is an estimate of the memory requirements for the solver
One should notice that all hard tables are already taken into account
and that only working tables are necessary

Figure 9.9: Working memory necessary to the calculation

The information regarding the memory effectively used ends the initialization phase.

The computation step itself is now treated. The convergence of the solver is now possible to print at each time step. From a general point of view, the present case seems to converge well. Indeed the residu (i.e the norm of the difference between the previous iteration is reduced at each iteration of the conjugate gradient algorithm. One may notice that 30 iterations are sufficient to reach the required precision of 10^{-6}. This relatively high number of iterations is linked to the relatively high time step (100 secondes) taken. Taking a smaller time step whould without any doubt reduce considerably the number of iterations necessary.

The following figures present the portions of the listing file corresponding to the time steps 1, 2 and 100. The printing level retained for this simulation has been set to (Normal printing level, or equal to 2)

```
+++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++   Time steps
               SOLID ITERATION NTSYR =      1  TEMPSS =  0.10000E+03                 Physical time (in seconds)
+++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++
                                                   CPU time spent since the start of the calculation
 ** CPU TIME BEFORE THE TIME STEP :    2.03000     (in seconds)

 ** CPU TIME BEFORE CONDUCTION RESOLUTION STEP :    2.03000

 *** GRCONJ: RESOLUTION BY CONJUGATE GRADIENT                      Convergence of the conjugate gradient
           ITERATIONS   RELATIVE PRECISION   ABSOLUTE PRECISION
               25           0.10813E-05       0.43251E-04
 GRCONJ  32 ITERATIONS    RELATIVE PRECISION =  0.17943E-07 ABSOLUTE PRECISION =  0.71773E-06

 -> Min temperature : 0.39986E+02 node   2797 -- Max temperature : 0.46238E+02 node   6944

 ** CPU TIME AFTER CONDUCTION RESOLUTION STEP :   2.73000      Maximum temperature and minimum temperature
                                                               reached on the domain at that present time
+++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++
               SOLID ITERATION NTSYR =      2  TEMPSS =  0.20000E+03
+++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++

 ** CPU TIME BEFORE THE TIME STEP :    2.73000

 ** CPU TIME BEFORE CONDUCTION RESOLUTION STEP :    2.73000

 *** GRCONJ: RESOLUTION BY CONJUGATE GRADIENT
           ITERATIONS   RELATIVE PRECISION   ABSOLUTE PRECISION
               25           0.27406E-06       0.10993E-04
 GRCONJ  30 ITERATIONS    RELATIVE PRECISION =  0.14442E-07 ABSOLUTE PRECISION =  0.57927E-06

 -> Min temperature : 0.40000E+02 node   6862 -- Max temperature : 0.47490E+02 node   6944

 ** CPU TIME AFTER CONDUCTION RESOLUTION STEP :   3.40000

                                         o
                                         o
                                         o
                                         o
================================================================================
=================   some 100 time steps later ==================================
================================================================================
                                         o
                                         o
                                         o
                                         o
+++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++
               SOLID ITERATION NTSYR =    100  TEMPSS =  0.10000E+05
+++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++

 ** CPU TIME BEFORE THE TIME STEP :   56.5600

 ** CPU TIME BEFORE CONDUCTION RESOLUTION STEP :   56.5600

 *** GRCONJ: RESOLUTION BY CONJUGATE GRADIENT
           ITERATIONS   RELATIVE PRECISION   ABSOLUTE PRECISION
 GRCONJ  23 ITERATIONS    RELATIVE PRECISION =  0.17669E-07 ABSOLUTE PRECISION =  0.73010E-06

 -> Min temperature : 0.40000E+02 node   1537 -- Max temperature : 0.49269E+02 node   9106
```

Figure 9.10: Information printed during each time step

The third block interesting to describe is located in the last part of the listing. It contains statistics on the calculation. In particular, it is where the memory effectively used throughout the calculation is given. One splits the CPU cost in two phases, the first one concerns the initialization steps, while the second one is relative to the computation itself.

```
          ================================================================
                  SYRTHES : SUMMARY OF THE CALCULATION

                           CONDUCTION :        100 TIME STEPS
                                             11688      NODES
                                              5688    ELEMENTS

                            RADIATION :          0 TIME STEPS
                                                 0      FACES
          ================================================================


          CPU TIME  (in seconds)
          =========

          INITIAL PHASE FOR THE CONDUCTION  . . . . . .       2.0300
          CALCULATION OF THE CONDUCTION . . . . . . . .      55.2700

          INITIAL PHASE FOR THE RADIATION . . . . . . .       0.0000
          CALCULATION OF THE RADIATION. . . . . . . . .       0.0000

          T O T A L   C P U   T I M E . . . . . . . . .      57.3000

          A V E R A G E   C P U   T I M E
            - CALCULATION OF THE CONDUCTION
                                       per time step . .       0.5527
                              per time step for 1000 nodes . .       0.0473
            - CALCULATION OF THE RADIATION
                                       per time step . .       0.0000
                                 per time step for 1 face . .       0.0000


          MEMORY SIZE (in million of integers or reals)
          =============

          HARD TABLES :
                   - integers . . . . . . . . . . . . . .       0.0461
                   - reals  . . . . . . . . . . . . . . .       0.1112
          WORKING TABLES :
              READING OF THE MESH
                   - integers . . . . . . . . . . . . . .       0.0000
                   - reals  . . . . . . . . . . . . . . .       0.1232
              INITIAL PHASE
                   - integers . . . . . . . . . . . . . .       0.0234
                   - reals  . . . . . . . . . . . . . . .       0.0000
              CALCULATION PHASE
                   - integers . . . . . . . . . . . . . .       0.0000
                   - reals  . . . . . . . . . . . . . . .       0.1671


          I N T E G E R   M E M O R Y  (IA) . . . . . . .       0.0695
              - indicated memory. . . .      0.4000
              - percentage used . . . .     17.3732 %

          R E A L   M E M O R Y (RA)  . . . . . . . . . .       0.2783
              - indicated memory. . . .      1.0000
              - percentage used . . . .     27.8317 %

          T O T A L   M E M O R Y (IA+RA) . . . . . . . .       0.3478
*                       - SYRTHES - END OF EXECUTION                     *
**********************************************************************************
```

Figure 9.11: Statistics given at the end of the run

Even, if care must always be taken regarding the CPU time, to give an idea, the presented calculation has been performed on a PC (so similar CPU time should also be found for small workstations). On large mainframe the CPU time expected whould be much smaller, especially for large configurations.

A quick look at the memory required may push to reduce the memory size indicated in the program *syrthes.F*. This allows a better use of the accessible computer ressources.

Likewise, the CPU time used in a previous calculation may help to set the proper number of time steps to get the results in the morning or see if a faster or more powerful computer is needed. A non normal CPU time may indicate problems relative to the degree of optimization of the compiler or the size of the problem compared to the computer used.

9.1.5.b The temperature field

At the end of the run, in the directory where the calculation has taken place, one should be able to find several files :

- the geometrical file whose name is here *3rond2d_geom*
- the result file whose name is here *3rond2d_resu1*
- the chrono file whose name is here *3rond2d_chrono1*
- the historic (or thermal probes results) file whose name is here *3rond2d_histo1*

The couple of files (*3rond2d_geom, 3rond2d_resu1*) and (*3rond2d_geom, 3rond2d_chrono1*) can be used to generate temperature contours thanks to post-processors like EnSight et Rubens). Utility programs are at disposal to transform these files into files having formats adapted to each post-processor (see 4.2.2).

The thermal probes results file displays results in several columns and can be used directly by numerous plotters (for example *gnuplot* et *xmgr*).

The temperature field obtained after 500 time steps is presented.

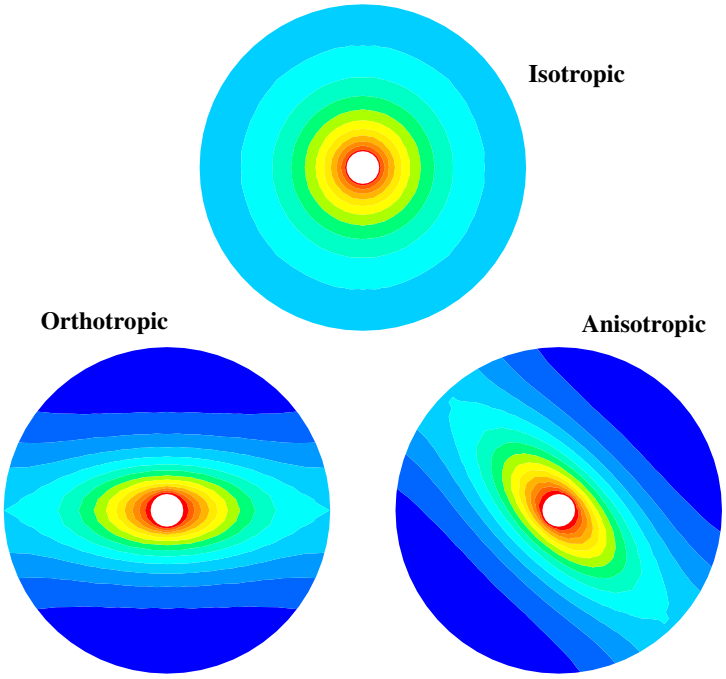

Figure 9.12: Temperature field obtained at t = 50000 s

It is interesting to note the behaviour difference between the different materials. The isothermals of the upper disk stay concentric, while for the others, ellipses appears before being affected by the outer boundary condition. One may underline that in the fully anisotropic case, the ellipses axis are not aligned with the axis of the reference system of coordinates.

9.2 How to use periodicity

9.2.1 Description of the case - Problem analysis

The aim of the present calculation is to simulate the heating of a large plate bored with a multitude of holes. Heat is brought at the inner side of the hole (for example by water flowing at high speed). The temperature is set at a constant temperature.
The dimension of the plate being considered as considerable, it may be interesting to consider the periodicity manifested by the plate and the boundary conditions.

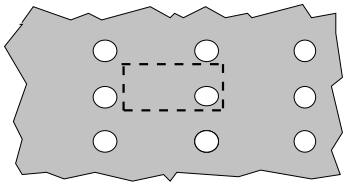

Figure 9.13: Detail of a large plate bored by plenty of holes

The restriction of the calculation domain to the periodic pattern allows to reduce considerably the calculation cost both in term of memory (number of nodes) and CPU. On the other hand, it obliges to take into account the periodicity along two directions.

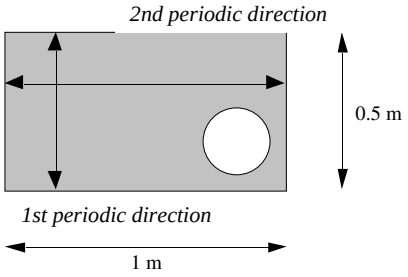

Figure 9.14: Calculation domain

If the vertical and horizontal periodicities do appear as fairly natural, one has also to deal with the corners of the domain. The temperature at a corner will be influenced by three other zones. This is illustrated by the following figure. For the lower left corner, the three domains to be accounted for the temperature calculation of this node have been represented.

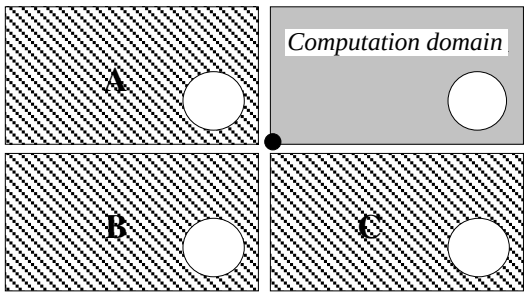

Figure 9.15: Calculation domain and fictive domains to be accounted for

Said otherwise, it means that for corner nodes, in case of double periodicity, the horizontal periodicity, the vertical periodicity, but also the diagonal periodicity have to be considered.

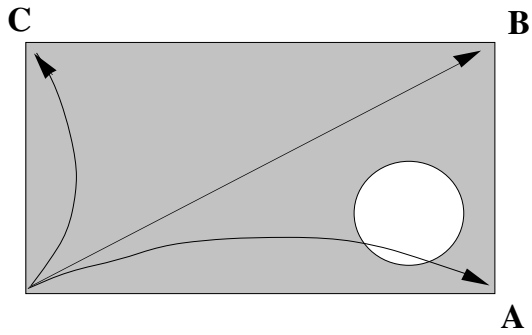

Figure 9.16: Cases of corner : periodicity along three directions

9.2.2 Generating the mesh

Generating the grid doesn't lead to particular problems. One has however to keep in mind the periodic phenomenon that one wants to simulate, especially regarding the references. The code requires to put the periodic faces into relation. One must therefore pay attention to identify the different edges and corners to be able to specify the periodic relation between them.
Moreover, a specific reference needs to be set on the inner side of the hole to put the proper boundary condition.

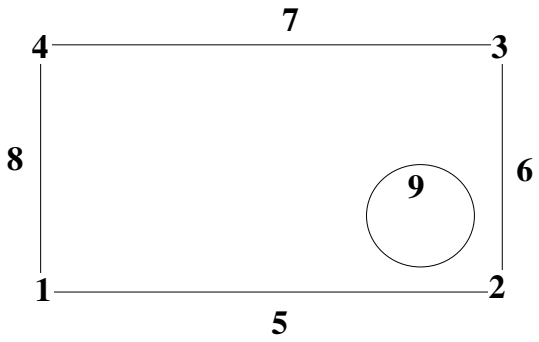

Figure 9.17: References on the border nodes

The mesh used contains 5788 nodes P_2 and 2798 elements. It is presented on the following figure.

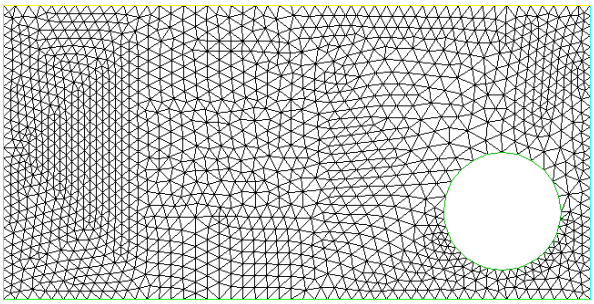

Figure 9.18: Mesh

9.2.3 Setting the parameter file

At the parameter file level, the specificity of the case is related to the periodic aspect. The data to define are gathered in the window **Boundary conditions**.
Initially, in that window, the section relative to periodicity has the following aspect : In the present case, one is going to use the periodicity of the kind translation along 2 directions. Regarding the border nodes affected by periodicity, one has to provide for each border, the references of the nodes belonging to the border 1 (Ref_1), the references of the nodes belonging to the border 2 (Ref_2), and the translation allowing to go from the border 1 to the border 2. Finally, one must not overlook the fact that the double periodicity induces that information must be given along the two diagonals of the domain.

```
'REFERENCES NOEUDS SOLIDES PERIODIQUES' 1 8 4 7 3 6 2 5

'CLIM'    'PERIODICITE'  'T'   1.  0.  0.   1 8 4   -1  2 6 3
'CLIM'    'PERIODICITE'  'T'   0.  0.5 0.   1 5 2   -1  7 3 4
'CLIM'    'PERIODICITE'  'T'   1.  0.5 0.       1   -1  3
'CLIM'    'PERIODICITE'  'T'   1. -0.5 0.       4   -1  2
```

To complete the boundary conditions, one sets an imposed temperature on the inner side of the hole (here $50°C$).

```
'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   9

'CLIM'    'DIRICHLET'      50     9
```

9.2.4 Results

The same comments could be done on the listing file and readers are encouraged to refer to the previous case *3rond2d*. The only differences are related to the periodicity option. From a calculation point of view, the taking into account of the double periodicity impacts very little the computing costs.

The following figure presents the temperature field obtained after 1200s of simulation.

In order to illustrate the periodic character of the simulation, the simulated domain has been duplicated 9 times. It is clear that an almost perfect match exists at the border of each sub-domain, leading to a continuous temperature field throughout the reconstituted global domain.

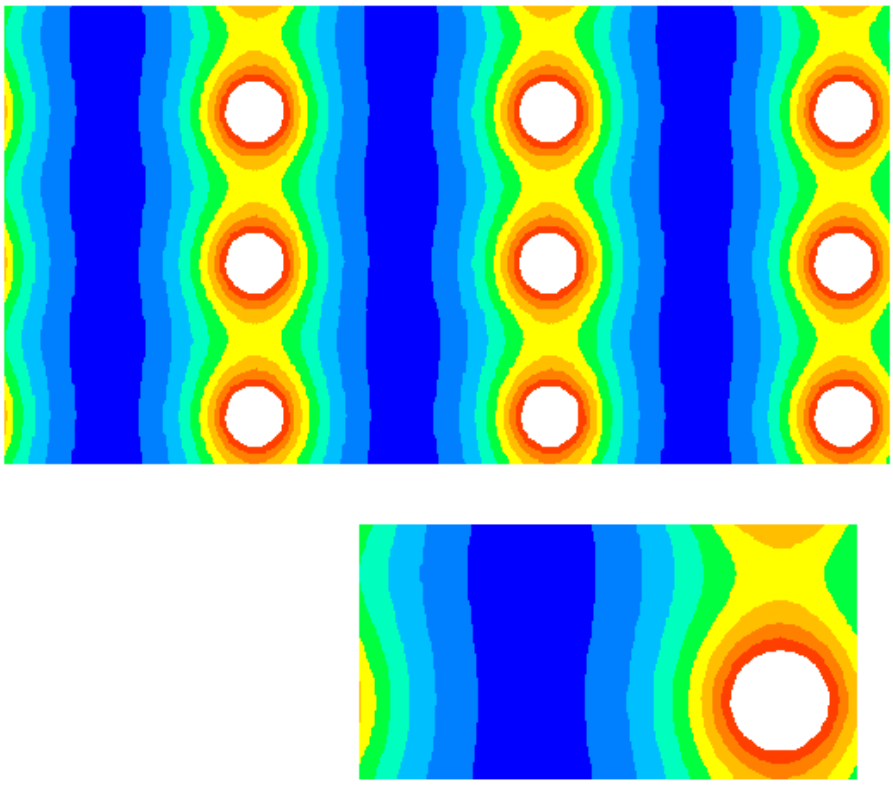

Figure 9.19: Temperature field at t = 1200 s

9.3 How to use contact resistance

9.3.1 Description of the case - Problem analysis

The aim of the present case is to illustrate the contact resistance aspects by the simulation of plate in which a crack exists. Initially the plate is at a uniform temperature of 20°C. One wishes to study the thermal transient when the upper part of the plate is subjected to a temperature of 50°C. The material is steel, the initial temperature is 20°C, and the contact resistance is fixed to 100 $W/m^2/K$.

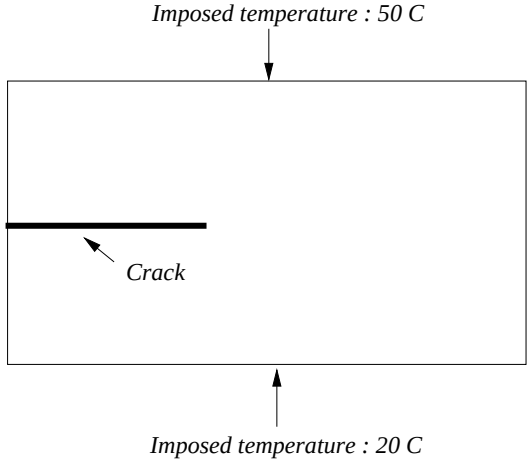

Figure 9.20: Sketch of the plate

9.3.2 Generation of the mesh

Special care has to be taken when dealing with contact resistance. Indeed one has to model the crack at the mesh level. From each side of the crack, nodes have to be considered as different even if they are located at the same place.

By zooming considerably the mesh near the crack, the following view should appear:

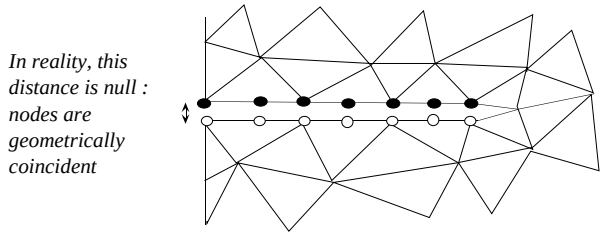

Figure 9.21: Zoom on the near crack mesh

One notes that the couple of nodes (•,◦) are geometrically coincident, but they have different node numbers. Indeed in the connectivity they belong to different elements, even if the nodes coordinates are exactly the same.

Moreover, the nodes • and ◦ need to carry different references since users will have to indicate that they are related by a contact resistance conditions. The references used in the present case are :

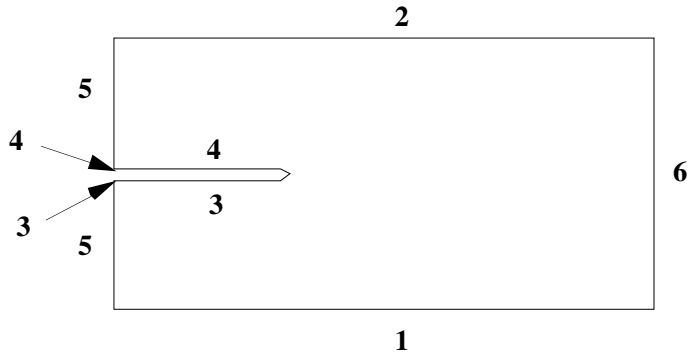

Figure 9.22: References used on the domain

The mesh contains 4887 P_2 nodes and 2798 elements.

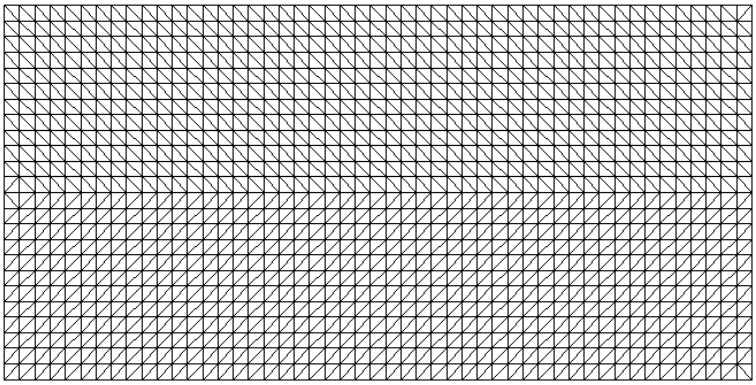

Figure 9.23: Mesh

9.3.3 Setting the parameter file

At the parameter file level, the contact resistance are considered as "simple" boundary conditions. One has simply to give the references of the nodes subjected to a contact resistance condition and what is the value of the contact resistance. These information have to be given in the window "Boundary conditions".

```
'REFERENCES NOEUDS OU FACES SOLIDES AVEC RESISTANCE DE CONTACT'  3 4

'CLIM'     'RES CONTACT'     100    3 4
```

The boundary conditions have to be completed by the setting of the temperature imposed on the upper side of the domain and on the lower side of the domain.

```
'REFERENCES NOEUDS SOLIDES AVEC DIRICHLET'   1 2

'CLIM'    'DIRICHLET'      20    1
'CLIM'    'DIRICHLET'      50    2
```

9.3.4 Results

The temperature fields obtained during the transient are displayed on figure 9.24. It is interesting to underline the discontinuity of temperature appearing at the crack level. At convergence the temperature discontinuity subsists, but the flux transfered stays of course continuous across the gap.

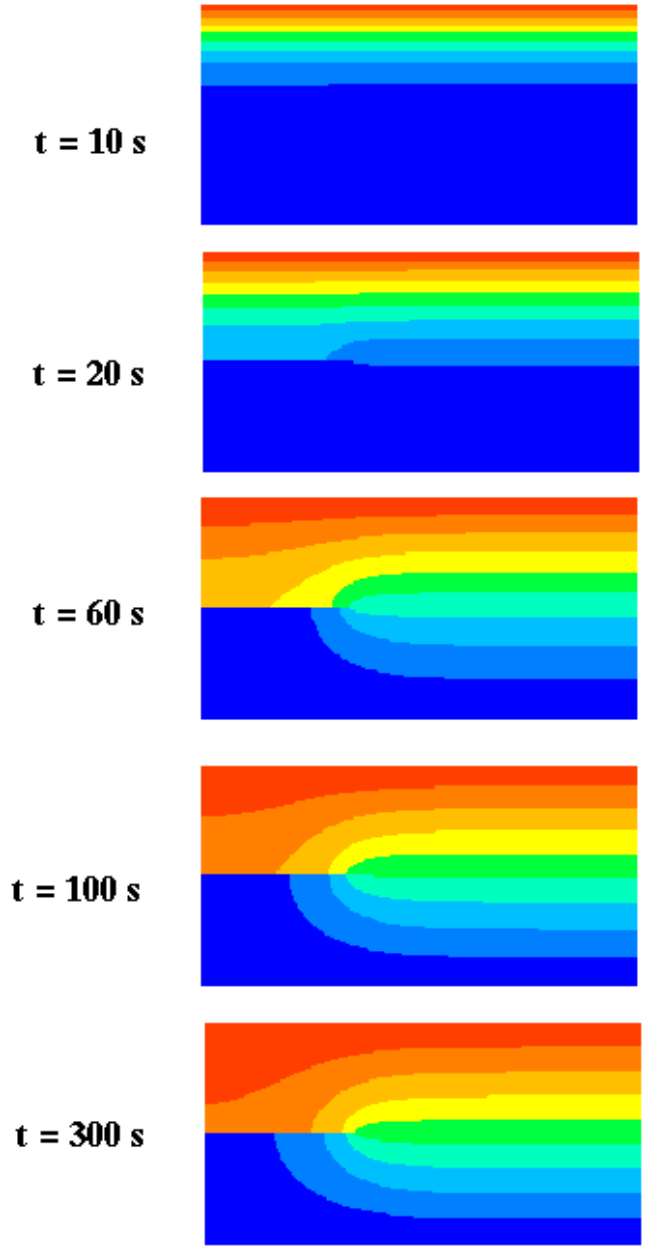

Figure 9.24: Temperature fields at different time during the transient

9.4 How to use the thermal radiation functionality

3rond_2d_ray

9.4.1 Description of the case - Problem analysis

The first illustrating case involving 3 disks is revisited. A circular enclosing envelop has been added to the calculation domain.

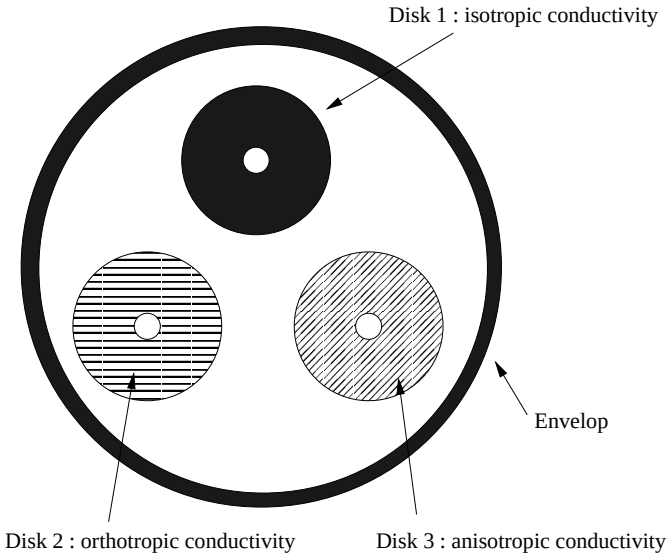

Figure 9.25: Sketch of the problem

- Disk 1 : the conductivity is the same along directions x or y (25 W/mK);
- Disk 2 : the conductivity along x is equal to 25 W/mK while it is only 5 W/mK along the y direction.
- Disk 3 : the conductivity is 25 W/mK and 5 W/mK along axis of a local system of coordinates. This local system of coordinates being at an angle of 45° with respect to the global system of coordinates of reference.
- Envelop : the conductivity is isotropic and equal to 25 W/mK.

The density and the specific heat are set respectively to : $\rho = 7700\ kg/m^3$ et $C_p = 460\ J/kgK$.

Regarding the emissivity, it is considered as identical for all pieces and constant throughout the spectrum.

9.4.2 Generation of meshes

This present calculation takes into account both radiative and conductive aspects, therefore two meshes have to be generated. The first one of dimension 2, will be used for conduction purposes, while the second one (dimension 1) will be used by for the radiation. Even if the geometry of the case stays simple from a meshing point of view, it is however clear that care has to be taken to set properly the references on the diverse parts of the domain. Indeed the simultaneous presence of several materials obliges to identify them clearly by different references. Likewise the border have to be referenced differently to be able to set different boundary conditions.

9.4.2.a The conduction grid

The problem is in dimension 2, and therefore made of 6 nodes triangles. This time it has been decided to identify the different materials through element references. Here the mesh generator being SIMAIL, 4 numbers (sub-domains colors for SIMAIL convention) have been used. One for each disk and a fourth one for the steel envelop. References on the border (edges) are also necessary to impose the proper boundary conditions (exchange and coupling with radiation).

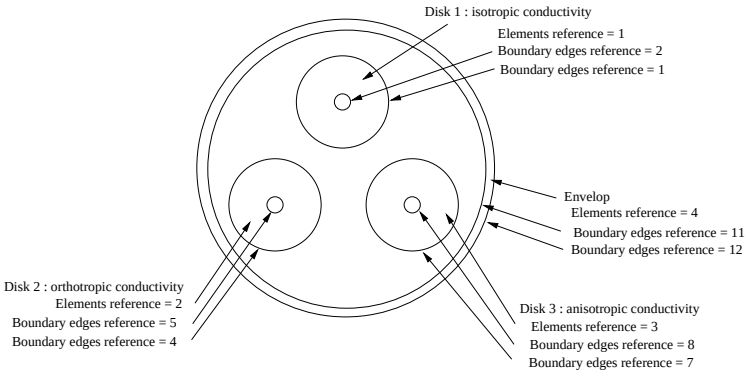

Figure 9.26: Setting the references

The mesh generated here contains 6504 P_2 nodes and 3072 triangles.

9.4.2.b The radiation grid

The case is in dimension 2, therefore the radiation grid is composed of 2 nodes segments. The heat transfer by radiation between the 3 disks and the envelops needs to be taken into account. The radiation grid will therefore be formed by the outside border of the disk and inner side of the protective envelop.

At the references level, the radiative properties being identical whatever the material, it is possible to use the same reference for the whole radiation mesh. There is no need to identify a sub-part to set a different emissivity for example. Here the reference chosen by the two authors is arbitrarily set to 1.

The radiation mesh contains 240 P_1 nodes and 240 elements (2 nodes segments).

9.4.3 Setting the parameter file

In the following paragraphs, the headings specific to this case will be presented. One will particularly insist on the radiation part and on the conduction/radiation coupling.

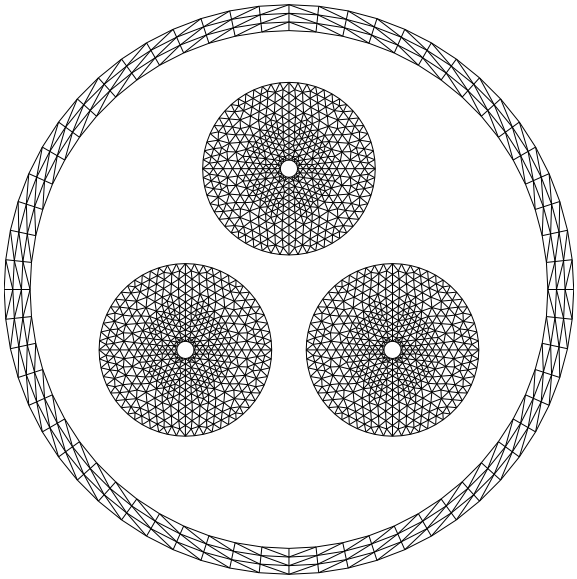

Figure 9.27: Conduction mesh

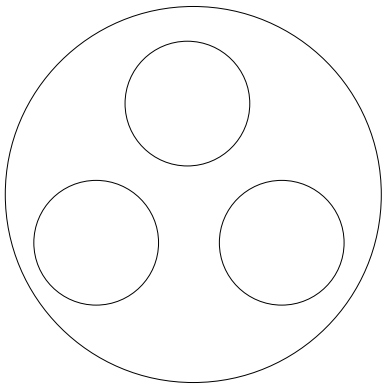

Figure 9.28: Radiation mesh

9.4.3.a Radiation

At the start, when users activate the radiation option, the window regarding the radiative aspects is looking as follows :
The first step consists in defining an internal point to each connex volume existing in the problem. One reminds that this information allows SYRTHES to identify the inner side from the outer side of the domain. Indeed, the surfacic radiation mesh issued from the mesh generator is generally not well oriented. Here, the radiation calculation taking place only between the three disks and the envelop, only one connex volume exists and thereafter, only one interior point is to be defined. The calculation domain has no symmetries or periodicity, and it is a closed enclosure.

```
'PERIODICITE DE ROTATION POUR LE RAYONNEMENT='  'NON'
'NOMBRE DE PLANS DE SYMETRIE POUR LE RAYONNEMENT=' 0
```

```
'DOMAINE DE RAYONNEMENT CONFINE OUVERT SUR L EXTERIEUR=' 'NON'

'RAYT'   'VOLUME CONNEXE'    -2.  -0.5  0.
```

Then physical properties have to be set. Here, only one spectral band is to be defined, and the emissivity is constant for all materials.

```
'NOMBRE DE BANDES SPECTRALES POUR LE RAYONNEMENT=' 1

'RAYT'  'BANDES SPECTRALES'     1     1.e-10      10.

'RAYT'  'EMISSIVITE PAR BANDE'  1     0.95       -1
```

9.4.3.b Radiation boundary conditions

In the present configuration, as it is often the case, all walls between which radiation takes place have been meshed for conduction. Therefore there is no need to apply boundary conditions directly on the radiation mesh. All radiation zones will be defined as being coupled to the conduction (see further).

9.4.3.c Radiation input/output

The options proposed by default are conserved. One may notice when running the case, that a small problem like this one do not induce long CPU time to calculate the view factors or build the correspondence data between the radiation mesh and the conduction mesh. Therefore, in this specific case, storing once for all these quantities in files does not appear has a major advantage. For much bigger configurations, one may have interest to compute these quantities only once and then activate the option "reading" from files.

```
'NIVEAU DES IMPRESSIONS POUR LE RAYONNEMENT=' 2
/
'NOMBRE DE REDECOUPAGES POUR CALCUL DES FACTEURS DE FORME=' 0
/
'STOCKAGE DES FACTEURS DE FORME SUR FICHIER=' 'NON'
'LECTURE DES FACTEURS DE FORME SUR FICHIER='  'NON'
/
'STOCKAGE DES CORRESPONDANTS POUR RAYONNEMENT='  'NON'
'LECTURE DES CORRESPONDANTS POUR RAYONNEMENT='  'NON'
```

9.4.3.d Files for the calculation

```
*
************************************************************************
*           EMPLACEMENT ET NOM DES FICHIERS DE SYRTHES                 *
************************************************************************
*
*
EMPLACEMENT DES FICHIERS POUR SYRTHES
  AMONT : ./
  SUITE : ./
  AVAL :  ./
*
*
```

```
NOM DES FICHIERS AMONT POUR SYRTHES
  DONNEES DU CALCUL : syrthes.data
  GEOMETRIE SOLIDE : 3rond2Dray.des
  DONNEES POUR LE RAYONNEMENT :  syrthes.ray
  MAILLAGE RAYONNEMENT : 3rond2D_r.des
*
*
NOM DES FICHIERS SUITE POUR SYRTHES
  SUITE SOLIDE RESU : resus1
*
*
NOM DES FICHIERS AVAL POUR SYRTHES
  RESU SYRTHES 1 : geoms
  RESU SYRTHES 2 : resus1
  CHRONO SYRTHES 2 : resusc1
  HISTORIQUE SOLIDE RESULTAT : histos1
  MAILLAGE DE RAYONNEMENT : ray.geo1
  RESULTATS DE RAYONNEMENT : ray.res1
  CHRONO DE RAYONNEMENT : ray.chro1
*
```

9.4.4 Coupling

This heading is aimed at defining the coupling between the conduction and the radiation phenomena. One must successively provide :

- on the conduction mesh : the list of references (on faces) coupled to radiation,
- on the radiation grid : the list of references (on faces) coupled to conduction.

Remark:
One speaks here of face references since it is the choice which has been made in the section "boundary conditions"

```
/ References sur le solide
/ ------------------------
'REFERENCES NOEUDS OU FACES SOLIDES AVEC RAYONNEMENT CONFINE' 1 4 7 11
/
/ References sur le maillage de rayonnement
/ -----------------------------------------
'RAYONNEMENT : REFERENCES FACES COUPLEES AU SOLIDE'  1
```

9.4.5 Results

9.4.5.a The listing file

Among all the possible output files, one may find either on screen or more reasonably in a file (if the standard output has been redirected), a list of information describing how the code is running.

We would like to remind that a "good" attitude is to have a close look at the listing file once the execution of code finished. As for a purely conductive study, this allows to check if errors have been introduced in the input data file or not. Finally, it helps figuring out if the code has behaved correctly.

We will not present again the listing file entirely, (refer to the case *3rond2d*), but only some information specific to configuration where radiation from wall to wall is activated.
The following figures present :
A summary of all the conditions set in the parameter file:

```
*** LECREF : REFERENCES OF SOLID NODES OR FACESWITH EXCHANGE
    2  5  8

*** LECRER : REFERENCES OF SOLID NODES COUPLED WITH RADIATION
    1  4  7 11

*** LECRER : RADIATION MESH
             REFERENCES OF FACES COUPLED WITH SOLID
    1  4  7 11

*** EVADIM : SOLID MESH CHARACTERISTICS :
             NUMBER OF ...
              - nodes                                    6504
              - volumetric elements                      3072
              - surfacic elements with flux               300
              - coupled faces                                0
              - coupled nodes                                0
              - faces with flux                              0
              - faces with Dirichlet                         0
              - faces with exchange                         60
              - faces with contact resistance                0
              - nodes with contact resistance                0
              - periodic volumetric elements                 0
              - periodic nodes                               0
              - surfacic elements with radiation           240
              - nodes with radiation                       480
              - faces with infinite radiation                0
              - nodes with rotation                          0
              - elements with volumetric flux                0

*** EVADIM : RADIATION MESH CHARACTERISTICS :
             NUMBER OF ...
              - radiation mesh - number of nodes           240
              - radiation mesh - number of elements        240
              - faces coupled with solid                   240
              - faces coupled with fluid                     0
              - faces with an imposed temperature            0
              - faces with an imposed flux                   0
              - faces with an 1D wall model                  0
              - nodes coupled with solid                   240
              - nodes coupled with fluid                     0
```

Figure 9.29: Checking of references and kind of boundary conditions

Then, some conduction geometrical information relative to the coupling with radiation:

```
*** XMAILL : SURFACIC SOLID MESH :
       - Number of surfacic coupled elements     :      0
       - Number of surfacic flux elements        :    300
       - Number of surfacic resistance elements  :      0
       - Number of surfacic radiation elements   :    240

*** LIMFNU : Solid, number of ...:
       - nodes coupled with fluid            0
       - faces coupled with fluid            0
       - faces with flux condition           0
       - nodes with Dirichlet condition      0
       - faces with exchange condition      60
       - nodes coupled with radiation      480
       - faces coupled with radiation      240
       - faces with infinite radiation       0
       - faces with contact resistance       0
       - nodes with contact resistance       0
       - periodic nodes                      0
       - moving nodes                        0
```

Figure 9.30: Geometrical information on the conduction mesh

Then SYRTHES gives a certain number of information on the boundary conditions set on the radiation grid:

```
*** LNRAY2 : RADIATION MESH
       - Number of nodes coupled with solid :    240
       - Number of nodes coupled with fluid :      0


*** LFRAY2 : RADIATION MESH
       - Number of faces with an imposed temperature :      0
       - Number of faces with an imposed flux        :      0
       - Number of faces with a 1D wall model and coupled with fluid  :      0
       - Number of faces with a 1D wall model (isolated faces) :      0
```

Figure 9.31: Boundary conditions on the radiation mesh

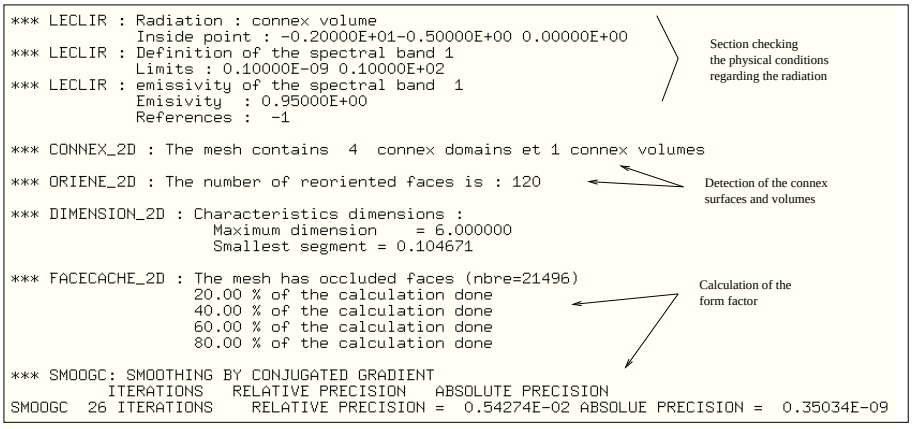

Figure 9.32: Preliminary radiation calculation

After this initial phase, one arrives to the calculation phase itself. Each time step is decomposed in a radiation problem followed by a conduction problem.

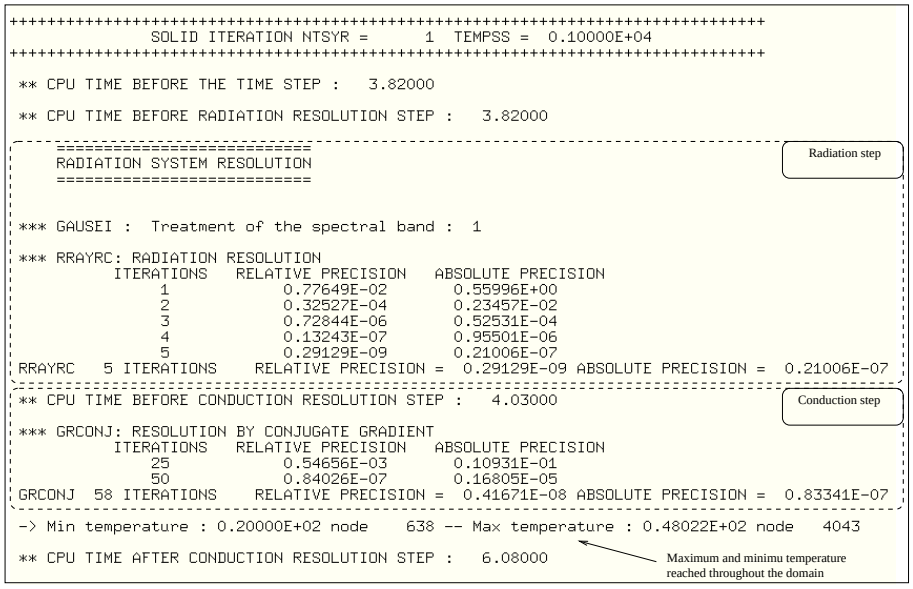

Figure 9.33: Informations given during each time step

Finally, at the end of the calculation, SYRTHES provides statistical information on the computation. In particular, one finds data relative to the core memory used for the initial phase and the calculation phase itself. These data distinguishes the conduction aspects from the radiation ones. Even if the information regarding the CPU time is often of limited interest, one may add that the log file given here corresponds to a calculation performed by the authors on a PC (unix system). The time step was 1000 seconds.

```
============================================================
        SYRTHES : SUMMARY OF THE CALCULATION

                CONDUCTION :         50 TIME STEPS
                                   6504      NODES
                                   3072   ELEMENTS

                 RADIATION :         50 TIME STEPS
                                    240      FACES
============================================================


CPU TIME  (in seconds)
=========

INITIAL PHASE FOR THE CONDUCTION  . . . . . .       1.6100
CALCULATION OF THE CONDUCTION . . . . . .          85.6800

INITIAL PHASE FOR THE RADIATION . . . . . .         2.2100
CALCULATION OF THE RADIATION. . . . . .             9.9600

T O T A L   C P U   T I M E . . . . . .            99.4600

A V E R A G E   C P U   T I M E
  - CALCULATION OF THE CONDUCTION
                                per time step . .   1.7136
                 per time step for 1000 nodes . .   0.2635
  - CALCULATION OF THE RADIATION
                                per time step . .   0.1992
                     per time step for 1 face . .   0.0008



MEMORY SIZE (in million of integers or reals)
=============

HARD TABLES :
         - integers . . . . . .                     0.0331
         - reals  . . . . . .                       0.0777
WORKING TABLES :
    READING OF THE MESH
         - integers . . . . . .                     0.0000
         - reals  . . . . . .                       0.0665
    INITIAL PHASE
         - integers . . . . . .                     0.0222
         - reals  . . . . . .                       0.0000
    CALCULATION PHASE
         - integers . . . . . .                     0.0000
         - reals  . . . . . .                       0.0925


I N T E G E R   M E M O R Y  (IA) . . . . . .      0.0553
    - indicated memory. . . .       0.1000
    - percentage used . . . .      55.3330 %

R E A L   M E M O R Y (RA)  . . . . . .            0.1702
    - indicated memory. . . .       0.5000
    - percentage used . . . .      34.0466 %

T O T A L   M E M O R Y (IA+RA) . . . . . .        0.2256
```

Figure 9.34: Final statistics at the end of the listing file

9.4.5.b The temperature field

The following figures display the temperature field obtained after 14 hours.

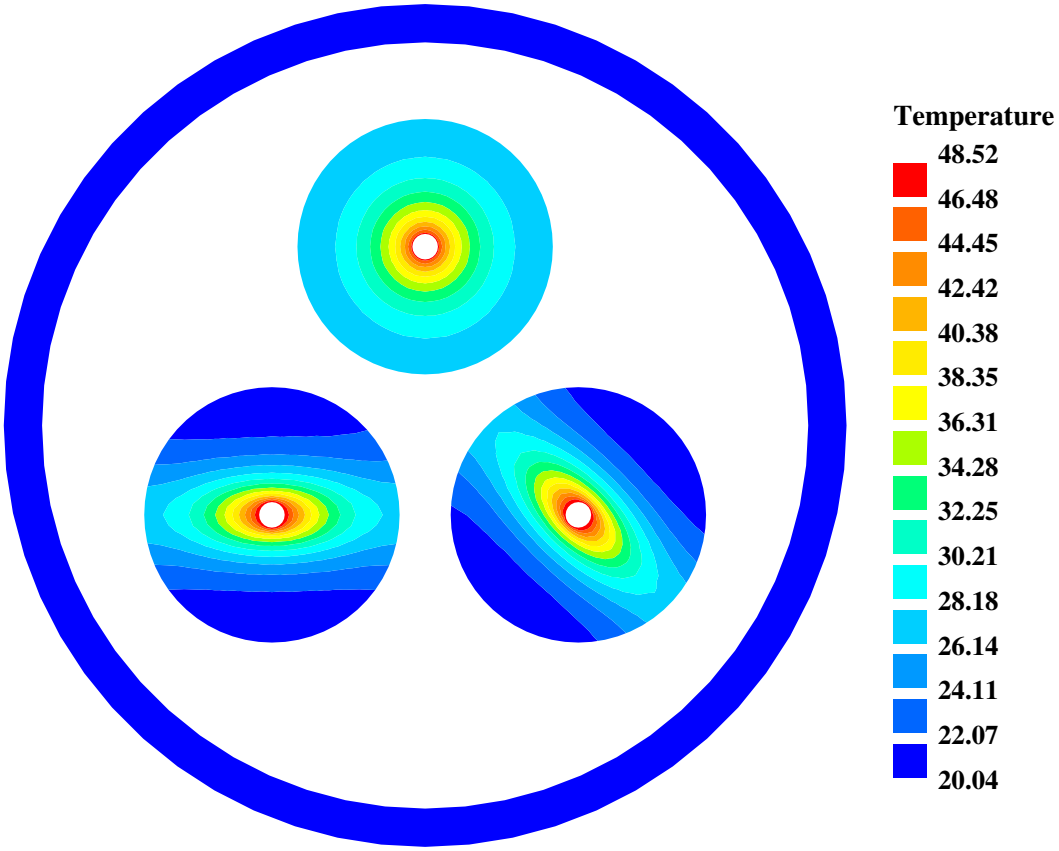

Figure 9.35: Temperature field in the solids at t = 14 hours

Again, the difference of behavior between the 3 disks is clearly noticeable. The isothermals of the upper disk stay concentric, while for the others, ellipses appear before being affected by the outer condition. One may underline that in the fully anisotropic case, the ellipses axis are not aligned with the axis of the reference system of coordinates.

Due to the linear scale used for the temperature contours, it is not easy to see the heating of the envelop coming from the heat transfer by radiation. The following figure uses a different scale more adapted to point out this effect.

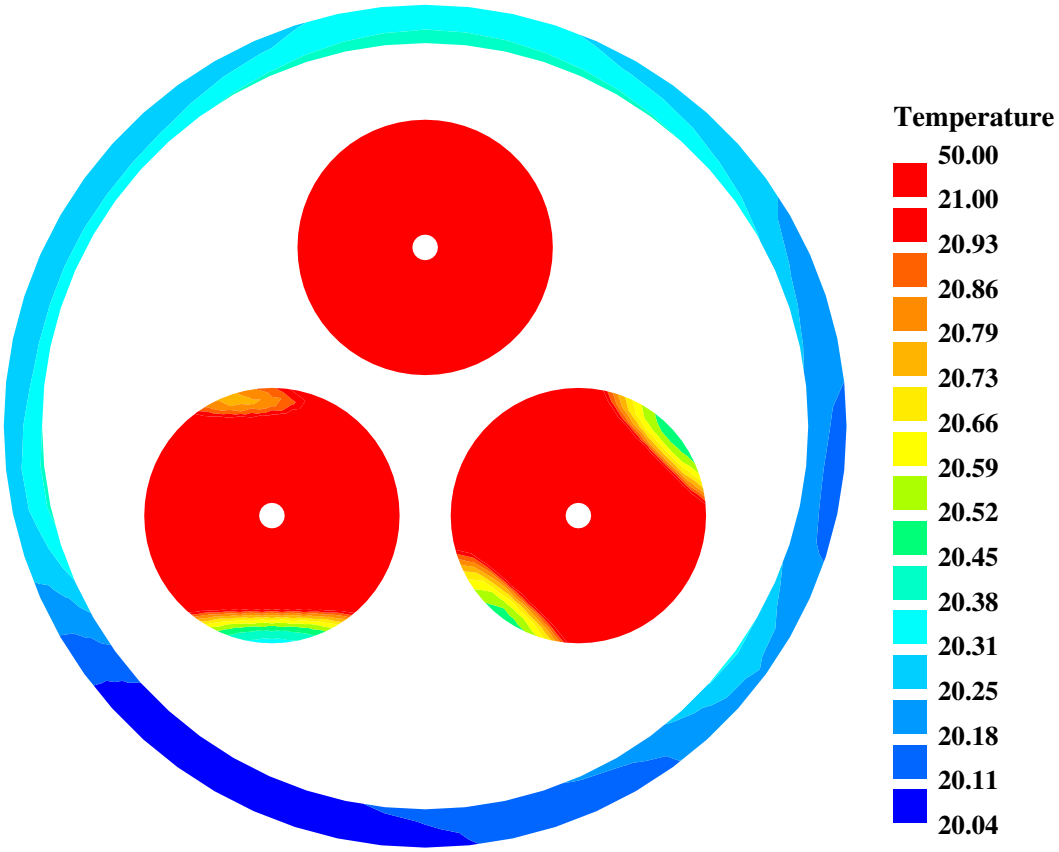

Figure 9.36: Temperature field in the solids at t = 14 hours

Conclusion 10

Today, SYRTHES can handle complex industrial problems, where conduction, radiation and convection (by coupling with a CFD code) phenomena are present simultaneously. One may hope that the more phenomena accounted for the more precise the simulation. However, one should also stay aware that it induces numerical simulation more difficult to manage. Indeed, in such global simulations, users have to tackle different physical phenomena with sometimes quite different time scales.

The elements gathered in this user manual should help users to get started with the code SYRTHES 3.4. The two authors of the SYRTHES code have always sought to decouple as much as possible the radiation phenomenon in order not to penalize users only interested by conduction studies. Moreover, it becomes easy to do a calculation including radiation and another without radiation to see the impact of this transfer mode on results. Indeed, radiation is too often neglected for wrong reasons, for example because users do not have access to numerical tools handling it correctly at low cost. In reality, it turns out to be often a major contributor in the heat transfer taking place, even when relatively low temperature are involved.

This manual presents also how to perform thermally coupled calculation with CFD code like *Code_Saturne*. Users have at their disposal a very complete numerical tool, allowing to study complex system including fluid/solid thermally coupled configurations. The almost total decoupling retained by the authors, should help users already familiar with *Code_Saturne*to learn how to use this additional feature at a fast pace.
For new users, it is easier to get started and split the learning in several independent modules. The time spent on one module can be completely reused afterwards. There is no need to master completely one CFD code in order to get started with the coupled version of the code SYRTHES, and likewise, users may start with a fluid code without any information on SYRTHES.
The same approach can be imagined in the treatment of particularly complex coupled systems. The tasks relative to the fluid and the solid can be dispatched among several engineers or teams.

Although possessing many features, the code SYRTHES will keep evolving, if users do have particular needs. In order to do so, a report page (the troubleshooting form) has been joined at the end of the document. The aim of the page is double. It is supposed to be a good way to describe the problems encountered, but also to describe possible improvements both in the present document, the version, or propose additional functionalities.

Physical quantities and units A

Quantity	Unité	Meaning
T	$^\circ C$	Temperature
ρ	kg/m^3	Density
C_p	$J/kg\,K$	Specific heat
k	W/mK	Thermal conductivity
ϕ	W/m^2	Surfacic flux
Φ	W/m^3	Volumetric flux
h	W/m^2K	Heat exchange coefficient
g	W/m^2K	Contact resistance
α_i	*degré*	Angles
ω	*rad/s*	Rotation velocity
$\vec{q}$	W/m^2	Fliux vector
ρ_i		Reflectivity
ε_i		Emissivity
F_{ij}		View factor
S_i	m^2	Surface (radiation)

SYRTHES file formats B

This chapter gives a detailed description of the formats used for the files issued from SYRTHES. Although some utility programs exist, allowing to transform results issued from SYRTHES, in formats accepted by some post-processors, users may have to read these result files (or a portion of them) to adapt the format to a special post-treatment, or not yet supported post-processors.

In this chapter, the characters strings appearing in the files will be detailed. The blank will be replaced by dots (.) in order to count or identify them more easily.

Apart the probes result file, which has a particular format, all results issued from SYRTHES, do follow the same structure. They are composed of two files, the first one corresponding to the geometrical aspects, while the second one contains the results (temperature and heat exchange coefficicient for the skin results) on the nodes of the mesh.

B.1 Description of the geometry

The files *geometry_solid*, *geometry_skin_fluid* and *geometry_radiation* are data bases containing respectively the meshes of the solid domain, the skin of the fluid mesh in contact with the solid, and the radiation mesh. They follow the format SYRTHES. They contain successively a list of nodes (with coordinates and references), a list of elements (with the connectivity) and optionally a list of face references. The geometrical file begins with the following type of header :

- lines 1 to 3 : comments
- line 4 : c··dimension·=·i1··element dimension·=·i1
- line 5 : c··number of nodes·=·i10
- line 6 : c··number of elements·=·i10
- line 7 : c··number of node by elements·=·i10
- line 8 : comment

The different section which follow may appear in any order in the file. Generally, one finds the nodes coordinates, then the connectivity of the mesh. Finally, in some cases, since this section is optional, the face references.
Each section is identified by a header:

The nodes coordinates :

- line 1 : c
- line 2 : c$·section·=·nodes
- line 3 : c

- line 4 à 3+number_of_nodes :

  C1, NUM, NUMREF,X,Y,Z

  ▷ C1 : character being "-" if it is a vertex, " " if it is a middle node.
  ▷ NUM : number of the node
  ▷ NUMREF : reference number of the node
  ▷ X Y Z : Nodes coordinates. One may notice that three coordinates are always present (in dimension 2, the third one stays unused; it is set to 0. (zero))

  The format used is : (I10,I3,1X,3(E14.7,1X))

The connectivity table :

- line 1 : C
- line 2 : C$·SECTION·=·ELEMENTS
- line 3 : C
- line 4 : 3+number_elements :

  NUM, NUMREF, LIST_OF_NODES

  ▷ NUM : element number
  ▷ NUMREF : reference number of the element (It should be noted that this quantity is not necessarily present and set to (0) zero in that case)
  ▷ LISTE_DES_NOEUDS : list of the nodes belonging to the element

  The corresponding format is (I10,I3,6I10) if the elements are 6 nodes triangles and (I10,I3,10I10) if the element are 10 nodes tetrahedra.

In that file the convention retained for the local numbering of the element is the following :

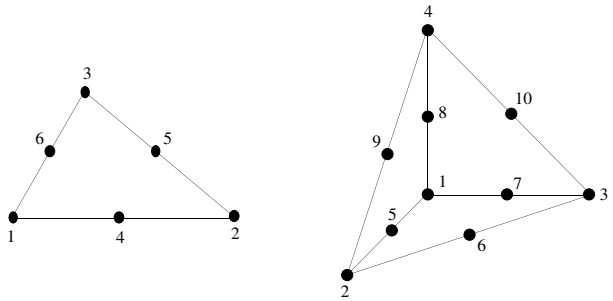

Figure B.1: Element local numbering convention in the geometrical result file

Table of face reference :

- line 1 : C
- line 2 : C$ SECTION·=·FACE REFERENCES
- line 3 : C

- line 4 to 3+number_of_elements :

  NUM, LIST_OF_REFERENCES

  or

  ▷ NUM : element number

  ▷ LIST_OF_REFERENCES : face references of the face belonging to the element

  The corresponding format is : (I10,1X,4I5)

One reminds here that in dimension 2, the reference of a "face" corresponds in reality to the reference given to each edge of the triangle :

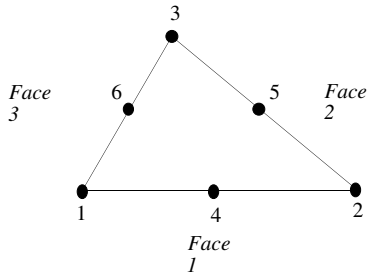

Figure B.2: "Faces" numbering in a triangle

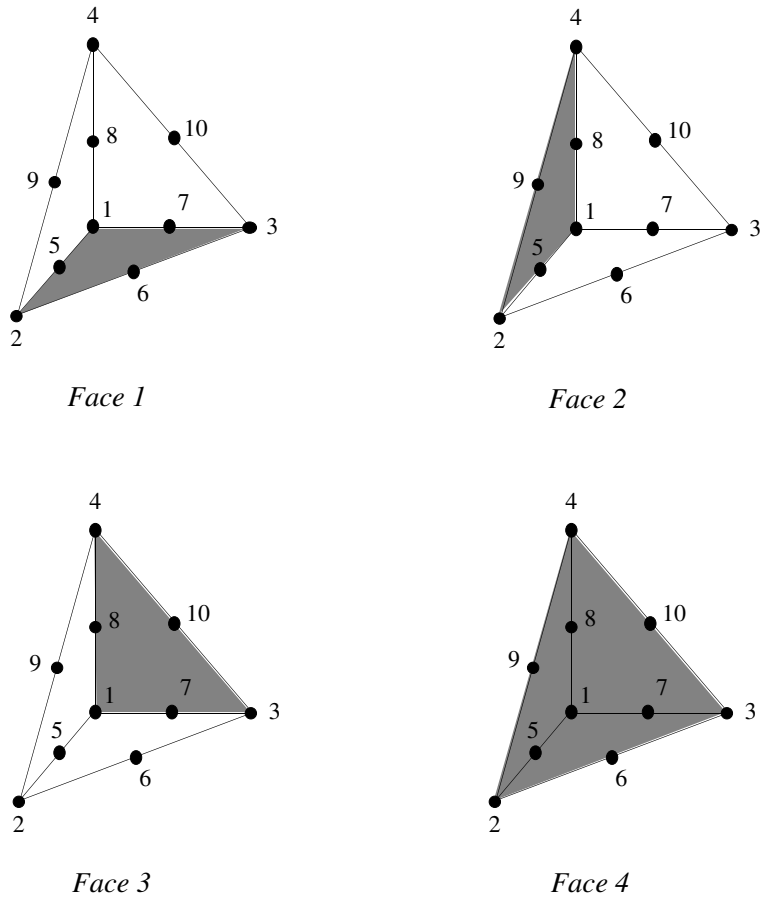

Figure B.3: "Faces" numbering in a tetrahedron

Here is an example of a part of a geometrical file :

```
C*********************************************C
C        FICHIER GEOMETRIQUE SYRTHES 3.4         C
C*********************************************C
C  DIMENSION = 3  DIMENSION DES ELTS = 3
C  NOMBRE DE NOEUDS =     514668
C  NOMBRE D'ELEMENTS =    362208
C  NOMBRE DE NOEUDS PAR ELEMENT =  10
C*********************************************C
C
C$ RUBRIQUE = NOEUDS
C
         1  0  0.2460071E+01  0.2019415E+01  0.0000000E+00
         2  0  0.2650000E+01  0.2019415E+01  0.0000000E+00
         3  0  0.2650000E+01  0.1887500E+01  0.0000000E+00
...

    514666  0  0.4347816E+00  0.5178590E-01  0.6088500E+00
    514667  0  0.4747987E+00  0.6250843E-01  0.6088500E+00
    514668  0  0.5220346E+00  0.6250843E-01  0.6088500E+00
C
C$ RUBRIQUE = ELEMENTS
C
         1 55     46465     48465     46466     46467     68443     68444     68445     68446     68447     68448
         2 55     46465     48464     46466     48465     68449     68450     68445     68443     68451     68444
         3 55     46465     48465     48463     48464     68443     68452     68453     68449     68451     68454
...
    362206  2     10137     12165     10029     10030    514659    514668    506899    506887    514662    506205
    362207  2     10029     12058     10030     12165    514273    514272    506205    514668    446956    514662
    362208  2     10029     12058     12165     12057    514273    446956    514668    291834    446275    446967
C
C$ RUBRIQUE = REFERENCES DES FACES
C
         1     0    0    0    0
         2     0    0    0    0
         3     0    0    0    0
...
    362206     0    0    0    0
    362207    57    0    0    0
    362208     0   57    0    0
```

B.2 Results at the nodes of the mesh

The code SYRTHES produces several result files. The final results are given in the result file. Users may also ask intermediate results to be stored in the "chrono" file.

In the case of a coupling with a CFD code, additional quantities located on the skin of the fluid mesh can also be available. Two additional files are then available (at the last time step, and intermediate time step). These file have the same structure as the classical result file. The results are constituted by the near wall fluid temperature, and the local heat exchange coefficient calculated by the CFD code.

In the case of calculation where radiation is present, specific results are available directly on the radiation mesh (temperature, flux). Then again the corresponding additional files are made available.

One reminds again that result file and "chrono" file have the same format, except the last one contains the results at several time steps.

Interfaces are available for several post-processors. Thus it is possible to visualize the results in the fluid, in the solid and on the skin mesh (see paragraph 4.2.2).

Description of the result file

Both result file and "chrono" file follow the same pattern. The only difference, is that the "chono" file contains several records. Each time step, is composed of a heading, then by the values of the current variable on each node of the mesh.
Heading of the time step :

- line 1 : comments
- line 2 : title of the calculation (A72)
- line 3 : comments
- line 4 : comments (4 characters then the list of the variable appearing at the next line)
- line 5 : CH, NDIM, NDIELE, NELEM, NBPN1, NBPN2, NBSCAL

  ▷ CH : string of 4 characters : C2C*
  ▷ NDIM : dimension of the problem (2 or 3)
  ▷ NDIELE : dimension of the elements (2=triangles, 3=tetrahedra)
  ▷ NBELEM : number of elements
  ▷ NBPN1 : number of nodes P_1
  ▷ NBPN2 : number of nodes P_2
  ▷ NBSCAL : number of scalars

  Format : (A4,6I8)

- line 6 : comments (4 characters then list of variables appearing at the next line)
- line 7 : CH, NDPT, TEMPS, DT

▷ CH : string of 4 characters : C4C*

▷ NDPT : number of the current time step

▷ TEMPS : physical current time (second)

▷ DT : value of the current time step (second)

Format : (A4,I8,2E16.6)

- line 8 : comments (4 characters then list of variables appearing at the next line)
- line 9 : CH, VERSION, DATE

  ▷ CH : string of 4 characters : C6C*

  ▷ VERSION : string of 8 characters indicating the version of the code used.

  ▷ DATE : string of 16 characters

  Format : (A4,A8,A16)

Then for each variable (1 in the case of the solid conduction and 2 for results on the fluid skin mesh or several according the number of spectral band for a radiation calculation) one finds the following part :

- line 1 : NOM_VARIABLE : string of 12 characters
- line 2 : I1,11('-')
  The interger indicates the kind of discretization of the current variable

  ▷ 1 : results on the mesh elements

  ▷ 3 : results on the mesh nodes
- following lines : results with the format : 6E13.7

A small part of result file is given below :

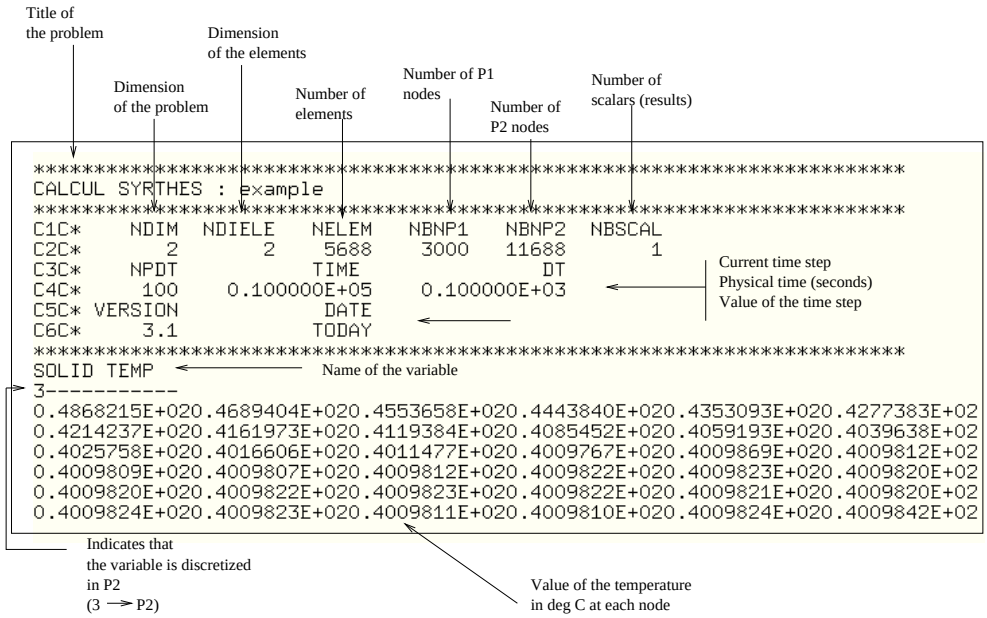

Figure B.4: Structure of a result file

B.3 Thermal probes results

The following figure indicates the format followed in the thermal probes results file. It should be noted that the example corresponds to a two dimensional case. In the case of a problem in dimension 3, the structures remains the same except that the coordinates of the nodes contain three coordinates instead of two.

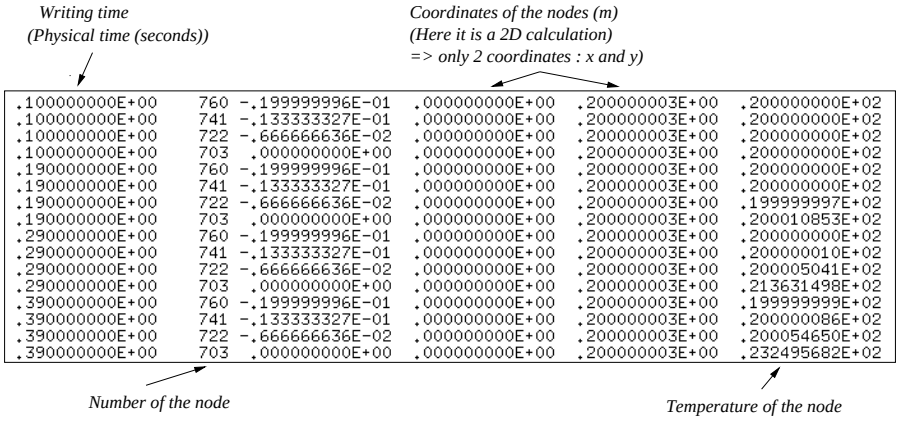

Figure B.5: Thermal probes result file issued from SYRTHES

The format of this file stays very simple and can be treated directly thanks to standard plotting softwares like *gnuplot* or *xmgr* available on Unix workstation or PC.

SYRTHES 3.4 user subroutines C

INITMP : **Initialization of the temperature field** **[page 73]**

By default, the initial temperature can be space dependent.

CPHYSO : **Material properties** **[page 73]**

It is possible to set precisely a function for :

- the density,
- the specific heat,
- the conductivity (isotropic, orthotropic or anisotropic).

By default, these properties are function of space, time, local temperature, etc... These can be set on nodes, elements or nodes by elements, according the option retained.

LIMSOL : **boundary conditions set on nodes** **[page 76]**

The boundary conditions can be of several types:

- Dirichlet,
- flux,
- exchange,
- contact resistance,

By default, these properties are space, time and local temperature dependent. This user subroutine is to be used if the option "boundary condition by nodes" has been activated.

LIMFSO : **boundary conditions set on faces** **[page 80]**

This subroutine is similar to LIMSOL, but corresponds to the case when "boundary condition by face" has been activated.

CFLUVS : **Volumetric flux** **[page 81]**

By default, the volumetric flux can be space, time, and local temperature dependent.

LIMRAY : **Radiation boundary conditions** **[page 83]**

Setting of the spectral bands, of the emissivity and complex boundary conditions on the faces of the radiation mesh.

INREFA : **Setting the references on element faces** **[page 82]**

Some mesh generators do not include the notion of face references. In the case users wish however to set the boundary conditions on faces and not on nodes, the user subroutine INREFA allows to build the faces reference table from the nodes references. It should be noted that this work may turn out to be long and somewhat tedious for very complex 3D geometries. Then the best option is maybe to use a mesh generator including this possibility directly.

Keywords SYRTHES 3.4 D

This paragraph provides the format of the keywords accepted by SYRTHES 3.4.

In the following table, the following convention is adopted :

- the column 'S' corresponds to the keyword use for SYRTHES calculations,
- the column 'S/F' corresponds to the keyword use for SYRTHES/CFD code calculation.
- the column 'Default' indicates the default value of the keyword,
- O : indicates it is compulsory to provide a value,
- $\nexists$: indicates that the keyword does not exist in that case,
- $\longrightarrow$ indicates that the value of the keyword is directly issued from the corresponding keyword given in the CFD code parameter file.
- — : indicates that the keyword is optional and has already a default value.

D.1 Conduction part : syrthes.data

D.1.1 Calculation parameters

Keyword	S	S/F	Default
'AXISYMMETRIC AXIS (NONE,OX,OY)='	—	→	none
'PROBLEM DIMENSION='	O	→	...
'BOUNDARY CONDITION BY NODE OR FACE='	—	—	node
'SETTING THE VOLUMETRIC FLUX BY='	—	—	element
'SETTING THE MATERIAL PROPERTIES BY='	—	—	element
'WRITING OF THE FLUID BOUNDARY MESH='	∄	—	no
'WRITING OF THE CHRONO FILE ON THE FLUID BOUNDARY MESH='	∄	—	no
'WRITING OF THE RESULT FILE ON THE FLUID BOUNDARY MESH='	∄	—	no
'FREQUENCY OF THE SOLID CALCULATION='	∄	—	1
'CONDUCTION PROBES='	—	—	no
'ISOTROPY OF THE MATERIAL='	—	—	1
'READING OF THE CORRESPONDENCE FILE='	∄	—	no
'PRINTING LEVEL FOR THE SOLID='	—	—	2
'NUMBER OF PERIODIC DIRECTIONS='	—	—	0
'NUMBER OF SOLID TIME STEPS='	O	→	...
'NUMBER OF ITERATIONS FOR THE SOLVER='	—	—	50
'FREQUENCY OF THE SOLID CHRONO FILE='	—	—	-1
'FREQUENCY OF THE SOLID LISTING FILE='	—	—	-1
'SOLID TIME STEP EQUAL TO FLUID TIME STEP='	∄	—	yes
'SOLID TIME STEP='	O	∄	...
'PRECISION CRITERIA FOR THE SOLVER='	—	—	10^{-6}
'ACTIVATION OF THE RADIATION COMPUTATION='	—	—	no
'SOLID IN TRANSLATION='	—	—	no
'CALCULATION RESTART='	—	→	...
'STORING OF THE CORRESPONDENCE FILE='	∄	—	No
'TITLE FOR THE SOLID CALCULATION='	O	O	...

D.1.2 Setting the references

'REFERENCES OF FLUID NODES COUPLED WITH SOLID'
'REFERENCES OF SOLID NODES OR FACES COUPLED'
'REFERENCES OF SOLID NODES WITH DIRICHLET'
'REFERENCES OF SOLID NODES OR FACES WITH FLUX'
'REFERENCES OF SOLID NODES OR FACES WITH EXCHANGE'
'REFERENCES OF SOLID NODES OR ELEMENTS WITH VOLUMETRIC FLUX'
'REFERENCES OF SOLID NODES OR FACES WITH CONTACT RESISTANCE'
'REFERENCES OF PERIODIC SOLID NODES'
'REFERENCES OF SOLID NODES IN ROTATION'
'REFERENCES OF SOLID NODES OR FACES WITH INFINITE RADIATION'

D.1.3 Physical conditions

```
/ ------------------------------
/ Setting the initial conditions
/ ------------------------------
/ 'CINI'    T(°C)                                                    references
/
/
/ ------------------------------------------------
/ Setting of the boundary conditions constant by block
/ ------------------------------------------------
/'CLIM'    'FLUX'     flux(W/m^2)                                    references
/'CLIM'    'DIRICHLET'     T(°C)                                     references
/'CLIM'    'EXCH COEF'     Text(°C) h(W/m^2/K)                       references
/'CLIM'    'CONTACT RES'     g(W/m^2/K)                              references
/'CLIM'    'PERIODICITY'    'T'    v_x v_y v_z                       references1 -1 references2
/'CLIM'    'PERIODICITY'    'R'    v_x v_y v_z      a1 a2 a3     b1 b2 b3
                                                                     references1 -1 references2
/'CLIM'    'RAYT INFINITE'    Text(°C) emissivity                    references
/
/'CVOL'    Φ(W/m^3)                                                  references
/
/ ------------------------------------------------
/ Setting the material properties constant by block
/ ------------------------------------------------
'CPHY'    'RHO'     ρ(kg/m^3)                                        references
'CPHY'    'CP'     C_p(J/kg/K)                                       references
'CPHY'    'K ISOTROPIC'     k(W/m^2/K)                               references
/
/ 'CPHY'    'K ORTHOTROPIC'    k_11 k_22 k_33                        references
/ 'CPHY'    'K ANISOTROPIC'    k_11 k_22 k_33    α_1 α_2 α_3         references
/
/
/ ------------------------------------------------
/ Setting boundary conditions specific to moving solids
/ ------------------------------------------------
/'SMOB'    'R'    v_x v_y v_z     α_1 α_2 α_3     β_1 β_2 β_3         references
/'SMOB'    'T'     v_x v_y v_z     vitesse                            -1
/'SMOB'    'ISOLATED NODES'     Text(°C) h(W/m^2/K)                   references
/'SMOB'    'INLET TEMPERATURE'     T(°C)                              -1
/'SMOB'    'EXTREMITIES REFERENCES'                                   references
/
/ -------
/ Thermal probes
/ -------
/'HIST'    'FREQ'    t
/'HIST'    'NODES'    n_1 n_2 n_3 ...


/ -------
/ Flux balance
/ -------
/'Surfacique flux balance'                                           references
/'Volumitric flux balance'                                           references
```

D.2 Radiation part : syrthes.ray

D.2.1 Calculation parameters

Keyword	Default
'RADIATION DOMAIN OPEN TOWARD THE EXTERIOR='	no
'RADIATION PROBES='	no
'PERIODICITY OF ROTATION FOR RADIATION='	no
'PRINTING LEVEL FOR RADIATION='	2
'NUMBER OF SPECTRAL BANDS FOR RADIATION='	1
'NUMBER OF SYMMETRY PLANES FOR RADIATION='	0
'NUMBER OF SPLITTING FOR THE VIEW FACTORS CALCULATION='	0
'FREQUENCY OF THE RADIATION CHRONO FILE='	-1
'STORING OF THE RADIATION CORRESPONDENCE FILE='	no
'STORING OF THE VIEW FACTORS FILE='	no
'READING OF THE RADIATION CORRESPONDENCE FILE='	no
'READING OF THE VIEW FACTORS FILE='	no

D.2.2 Setting of the references

/ References on the solid
/ ————————————
'REFERENCES OF SOLID NODES OR FACES COUPLED WITH RADIATION'
/
/ References on the radiation mesh
/ ————————————————————
'RADIATION : REFERENCES OF THE COUPLED WITH SOLID FACES'
'RADIATION : REFERENCES OF THE IMPOSED TEMPERATURE FACES'
'RADIATION : REFERENCES OF THE IMPOSED FLUX FACES'

D.2.3 Physical conditions

/ ————————————————————————————
/ Setting of the radiation parameters
/ ————————————————————————————
/'RAYT' 'SYM3D' ax + by + cz + d = 0
/'RAYT' 'SYM3D' 0. 0. 1. -0.5
/
/'RAYT' 'SYM2D' ax + by + c = 0
/'RAYT' 'SYM2D' 0. 1. -0.5
/
/ 'RAYT' 'CONNEX VOLUME' P_x P_y P_z
/
/'RAYT' 'PERIO3D' $P_xP_yP_z$ $A_xA_yA_z$ α
/
/'RAYT' 'PERIO2D' P_xP_y α
/
'RAYT' 'SPECTRAL BANDS ' $\lambda_1\lambda_2$

/ bande emissi ref

'RAYT' 'EMISSIVITY BY BAND' num_bande emissivity references
/
/'RAYT ' 'INFINITE TEMPERATURE' Temperature($^\circ C$)
/
/
/ ——————————————————————————-
/ Radiation boundary conditions
/ ——————————————————————————-
/'RAYT' 'IMPOSED TEMPERATURE' température ($^\circ C$) références
/
/'RAYT' 'IMPOSED FLUX BY BAND' bande flux(W/m^2) références
/
/ ———
/ Radiation probes
/ ———
/'PROBES' f_1 f_2 f_3 ...

Troubleshooting form for SYRTHES E

This new version of the code SYRTHES, extends considerably the possibility of the code. In particular the radiation aspects (transparent medium) are now available.

If users are confronted with troubleshooting, it is crucial to fill out the following form. Only this effort will lead to get an adequate answer, and if possible a solution to overcome the problem. Each form received by the support team of the code SYRTHES, will be carefully analyzed and a conclusion sent back to the user having emitted the form.

This form can also be a privileged way to inform the developers of the users wishes regarding the existing functionalities. It can also be used to ask for new functionalities to be developed and implemented in SYRTHES.

SYRTHES form

Name : Telephone : Date :

Company : e−mail :

Address : Computer :

Release of SYRTHES ☐

Option used	*Kind of error*	*Activation of the radiation option*	*Problem located in*
☐ SYRTHES	☐ IMPROVEMENT	☐ YES	☐ CONDUCTION
☐ SYRTHES / ESTET	☐ BLOCKING PROBLEM	☐ NO	☐ RADIATION
☐ SYRTHES / N3S	☐ NON BLOCKING PROBLEM		

General description of the conduction problem

Dimension	*Boundary conditions*	*Conductivite*	*Volumetric flux*
2D	☐ DIRICHLET	☐ ISOTROPIC	☐ YES
2D axi	☐ FLUX	☐ ORTHOTROPIC	☐ NO
3D	☐ EXCHANGE	☐ ANISOTROPIC	
	☐ INFINITE RAD		

Periodicity	*Moving solid*	*Sequel calculation*	*User subroutines list*
☐ AUCUNE	☐ NON	☐ OUI	
☐ TRANSLATION	☐ TRANSLATION	☐ NON	
☐ ROTATION	☐ ROTATION		

Description of the radiation calculation

Boundary conditions	*Presence of*	*Number of spectral bands*
☐ COUPLED TO A SOLID	☐ SYMMETRY	☐ ONLY ONE
☐ TEMPERATURE IMPOSED	☐ PERIODICITY	☐ SEVERAL
☐ FLUX IMPOSED		
☐ ISOLATED EQUIV WALL		
☐ EQUIV WALL COUPLED TO FLUID		

Description of the problem or of the improvement

Return to : EDF R&D (6 quai Watier −78401 Chatou Cedex)

Bibliography

[1] Péniguel C. - *Delft.* Heat Transfer Simulation for Industrial Applications : Needs, Limitations, Expectations 2nd Int Conf on Turbulence Heat and Mass Transfer - Delft.

[2] Sakiz M. *Guide pratique d'utilisation de Code_Saturne Version 1.0.* Rapport EDF HI-83/01/015/A.

[3] *Manuel théorique de Code_Saturne Version 1.0.* En cours de rédaction.

[4] Péniguel C, Rupp I. *Couplage Thermique Fluide-Solide - Version 1.0 - Résolution des équations de la chaleur au sein d'un solide en éléments finis (Théorie - Méthodes Numériques - Validation).* Rapport EDF/DER HE-41/93/015/A.

[5] Péniguel C, Rupp I. *SYRTHES - Conduction et rayonnement thermique - Manuel théorique de la version 3.3.* Rapport EDF R&D HI-83/01/039/A.

[6] Rupp I, Péniguel C. *SYRTHES - Conduction et rayonnement thermique - Manuel de validation de la version 3.3.* Rapport EDF R&D 2002.

[7] Péniguel C, Rupp I. A numerical method for thermally coupled fluid and solid problems. pages 1027–1039, U.K. Swansea, 1993. Numerical Methods in Thermal Problems.

[8] Péniguel C, Rupp I. A finite element approach to simulate general conduction problems. pages 555–562, U.K. Southampton, 1994. 3rd Int Conference Heat Transfert.

[9] Beaucourt D, Chraibi H. *Calcium V2 : guide d'utilisation.* Rapport EDF/DER HI-76/96/009/A.

[10] Péniguel C , Rupp I. Coupling conduction radiation and convection using PVM. St Venant Symp. Paris, 1997.

[11] Geist A, Beguelin A, Dongarra J, Jiang W, Mancheck R. PVM 3 user's guide and reference manual. Oak Ridge Tennesse 37831. Oak Ridge National Laboratory.

[12] Péniguel C, Rupp I. A numerical approach for thermally coupled fluid and solid problems in complex geometries. pages 27–34, U.K. Southampton, 1994. 3rd Int Conference Heat Transfer.

[13] Rupp I , Péniguel C. Coupling heat conduction and radiation and convection phenomena in complex 2D and 3D geometries. U.K. Swansea, 1997. Numerical Methods in Thermal Problems.

[14] *Documentation technique du logiciel SIMAIL.* SIMULOG.

[15] SDRC. *Logiciel Ideas MS - I-deas Finite element modeling - maillage (supertab),Aide en ligne du logiciel.*

[16] CEI. *EnSight - Users guide.* CEI - 1997.

[17] Piro M. *Rubens : Manuel de référence version 2.0.* Rapport EDF HE-45/92.01.

[18] Berthou J.Y., Lefebvre V. *Guide d'utilisation de la bibliothèque d'échanges de données MED V2.0.* Rapport EDF/DER HI-76/2000/002/A.

[19] Berthou J.Y., Lefebvre V. *Guide de référence de la bibliothèque d'échanges de données MED V2.0.* Rapport EDF/DER HI-76/2000/003/A.

Index